

The Advanced Pocket Professional™

Spice48

Owner's Manual

SPARCOM®

Edition 1
Manual Reorder No. 11511-A

February 1992
Software Reorder No. 10511-1A

The Pocket Professional®

Spice48

Owner's Manual

SPARCOM CORPORATION

Edition 1
Manual Reorder No. 11511-A

February, 1992
Software Reorder No. 10511-1A

Notice

Sparcom Corporation shall not be liable for any errors, loss of profit, or any other commercial damage, including but not limited to special, consequential, incidental or other damages, resulting from or in any way connected with the furnishing, performance, or use of this manual or software. This manual, the accompanying software, and the examples contained herein are provided "as is" and are subject to change without notice. **Sparcom disclaims any other warranties, expressed or implied, including but not limited to the implied warranties of merchantability and fitness for a particular purpose.** (For more information about warranty, refer to Appendix A, "Warranty and Service.")

This manual and the accompanying software are © Sparcom Corporation 1992. All rights reserved. Reproduction, transmission in any form or by any means, adaptation, or translation of this manual is prohibited without prior written permission of Sparcom, except as allowed under the copyright laws.

The owner of this manual is granted a one-user, non-commercial license to use the enclosed software, and may not copy, distribute, or transfer the software under any circumstances without prior written permission of Sparcom.

Portions of this manual are © Hewlett-Packard Company 1990 and are used with permission.

Portions of this software are licensed from Armstrong Publishing, Corvallis, OR, U.S.A.

Sparcom wishes to express appreciation to Chris Wolfe, Dr. Mark Yoder, and especially to Dr. Dave Voltmer, of Rose-Hulman Institute of Technology, for their conception of the product and extensive efforts in its development.

Pocket Professional and Sparcom are registered trademarks of Sparcom Corporation.

Sparcom Corporation
897 NW Grant Avenue
Corvallis, OR 97330
U.S.A.

Printing History

Edition 1

February 1992

Contents

1	Getting Started	9
	Installing and Removing a ROM Card	9
	Installing a ROM Card	10
	Removing a ROM Card	10
	Starting the Spice48 Pac	11
	Moving Around the Screen	12
	Using the Main Menu	12
	Items in the Main Menu	12
	Summary of Operations	13
	Changing the Font Size	14
	Viewing Items Too Wide for the Display	14
	Using the Search Mode	14
	Text Editing	15
	Summary of Operations	15
	Alpha Lock	16
	How to Load Data from the Stack	16
	System Flags	16
	User Flags	17
	Memory Requirements	17
	The 'SPARCOM' Directory	18
2	Spice48 Tutorial	19
	Using the Spice48 Tutorial	19
	Example: RC Circuit	19
	Adding the Circuit RC	20
	Adding Nodes to RC	21
	Adding Elements to RC	22
	Drawing the Schematic of RC	30
	Analyzing RC	33
	Example: Bipolar Junction Transistor, DC Model	42
	Adding the Circuit BJTDC	43
	Adding Elements and Nodes to BJTDC	44

Drawing the Schematic of BJTDC	50
Analyzing BJTDC	53

3 Circuit Manager 61

Using the Circuit Manager	61
Items in the Circuit Manager	61
Summary of Operations	62
Selecting the Current Circuit	63
Adding a Circuit	63
Deleting a Circuit	64
Packing a Circuit.....	64
Transferring a Circuit	64
Transferring a Circuit to a Computer	64
Transferring a Circuit from a Computer	65

4 Element Manager 67

Using the Element Manager	67
Items in the Element Manager	68
Adding an Element	69
Deleting an Element	69
Editing an Element	69
Element Parameter Entry Tips.....	70
Selecting a Node during Parameter Entry	70
Selecting an Element during Parameter Entry	71
Summary of Operations	71
R–Resistor.....	72
C–Capacitor	73
L–Inductor	74
K–Coupled (Coupled Inductors).....	75
V–Source (Independent Voltage Source)	76
I–Source (Independent Current Source)	77
E–VCVS (Voltage-Controlled Voltage Source)	78
F–CCCS (Current-Controlled Current Source)	79
G–VCCS (Voltage-Controlled Current Source)	80
H–CCVS (Current-Controlled Voltage Source)	81
Transient Sources	82
Pulse Transient	83
PWL Transient.....	84
Function Transient.....	85

5 Analysis Manager87

Using the Analysis Manager	87
Items in the Analysis Manager	88
Summary of Operations	88
Preparing the Current Circuit	89
Analysis Parameter Entry Tips	90
Selecting a Node during Parameter Entry	91
Selecting an Element during Parameter Entry	92
Summary of Operations	92
Operating Point (OP) Analysis	93
Transfer Function (TF) Analysis	94
DC Sweep (DC) Analysis	95
AC Sweep (AC) Analysis	96
Transient Sweep (TR) Analysis	97
New TR Analysis	98
Continuing TR Analysis	99

6 Node Manager 101

Using the Node Manager	101
Items in the Node Manager	101
Summary of Operations	102
Adding a Node	103
Deleting a Node	103
Editing a Node	103
Setting the Reference Node	104

7 Graphics Manager 105

Using the Graphics Manager	105
Items in the Graphics Manager	106
Summary of Operations	106
Creating an Accurate Schematic	107
Coordinate Entry Tips	108
Setting Node Coordinates	108
Summary of Operations	109
Setting Element Coordinates and Orientations	110
Summary of Operations	111
Setting Wire Coordinates	112
Summary of Operations	113
Viewing and Printing the Schematic	114
Viewing the Schematic	114

Printing the Schematic	115
Updating a Schematic	115
Clearing All Graphics	115

8 Data File Manager 117

Using the Data File Manager.....	117
Items in the Data File Manager.....	117
Summary of Operations	118
Loading a Data File	119
Deleting a Data File	119
Transferring a Data File.....	119
Transferring a Data File to a Computer	119
Transferring a Data File from a Computer	120

9 Output Manager 121

Using the Output Manager	121
Understanding Data Slices	122
Output Manager Views.....	123
Data View	123
Understanding the Data View	124
Using the Data View	126
Summary of Operations	127
Node View	128
Using the Node View	128
Summary of Operations	130
Source Value View	131
Using the Source Value View	131
Summary of Operations	133
Frequency View	134
Using the Frequency View	134
Summary of Operations	136
Time View	137
Using the Time View	137
Summary of Operations	139
Node Plots.....	140
Plotting Ranges	140
Node Plot.....	140
Summary of Operations	141
Value Plots.....	141
Plotting Ranges	142
Point Plot.....	142
Line Plot	143

Equation Plot	143
Summary of Operations	144
Display Forms	145
Selecting Display Type	145
Selecting Display Precision	146

10 Miscellaneous Info 147

Using Miscellaneous Info.....	147
Items in Miscellaneous Info	148
Summary of Operations	148
Resistor Colors	149
Resistor Band Descriptions and Restrictions	149
Summary of Operations	150
Constant Library.....	151
Summary of Operations	151
GaAs and Si Properties	152
Standard Components	152
Standard Prefixes	153

A Warranty and Service 155

Pocket Professional™ Support	155
Limited One-Year Warranty	156
What is Covered.....	156
What is Not Covered	156
If the Card Requires Service	156
Service Charge	156
Shipping Instructions	157
Environmental Limits.....	157

B Summary of Operations 159

Summary of Screen Softkeys	159
Summary of Softkey Actions	161

C Index 165

Chapter 1

Getting Started

Sparcom's Advanced Pocket Professional™ software is the first of its kind, developed to provide speed, efficiency and portability to students and professionals in the technical fields. When you slide the Advanced Pocket Professional™ Spice48 Pac into your HP 48SX, your calculator is instantly transformed into an electronic "textbook," ready to efficiently solve your electrical circuit problems. The Spice48 Pac is organized into eight separate modules: the Circuit Manager, Element Manager, Analysis Manager, Node Manager, Graphics Manager, Data File Manager, Output Manager, and Miscellaneous Info... all available in an efficient, menu-driven format.

This chapter covers:

- ☐ Installing and Removing a ROM Card
- ☐ Moving Around the Screen
- ☐ Using the Main Menu
- ☐ Changing the Font Size
- ☐ Viewing Items Too Wide for the Display
- ☐ Using the Search Mode
- ☐ Text Editing
- ☐ Alpha Lock
- ☐ How to Load Data from the Stack
- ☐ System Flags
- ☐ User Flags
- ☐ Memory Requirements
- ☐ The 'SPARCOM' Directory

Installing and Removing a ROM Card

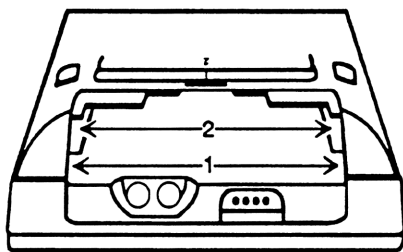
The HP 48SX has two ports for installing plug-in ROM cards. You can install your Spice48 Pac in either port.

WARNING: Turn off the HP 48SX while installing or removing a ROM card! Otherwise, your HP 48SX memory may be erased.

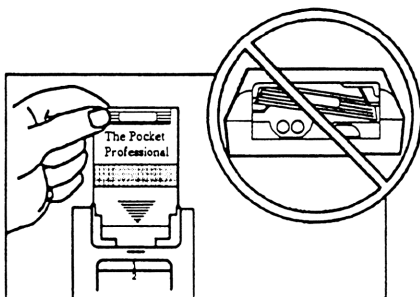
Installing a ROM Card

To install a ROM card, follow these steps:

- ❶ Turn the HP 48SX off. Do not press **[ON]** until you have completed the installation procedure.
- ❷ Remove the port cover. Press against the grip lines and push forward. Lift the cover to expose the two plug-in ports, as shown below:



- ❸ Select either empty port for the Advanced Pocket Professional™ card, and position the card just outside the slot. Point the triangular arrow on the card toward the HP 48SX port opening, as shown below:

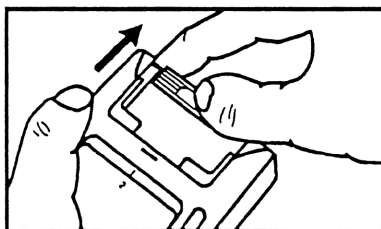


- ❹ Slide the card firmly into the slot. After you first feel resistance, push the card about 1/4 inch further, until it is fully seated.
- ❺ Replace the port cover.

Removing a ROM Card

To remove a ROM card, follow these steps:

- ❶ Turn the HP 48SX off. Do not press **ON** until you have completed the removal procedure.
- ❷ Remove the port cover. Press against the grip lines and push forward. Lift the cover to expose the two plug-in ports, as shown above.
- ❸ Press against the card's grip and slide the card out of the port, as shown below:

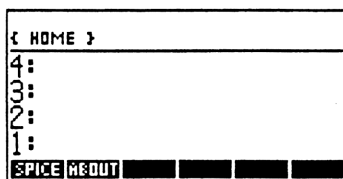


- ❹ Replace the port cover.

Starting the Spice48 Pac

After you turn on your HP 48SX by pressing **ON**, there are three ways to start the Spice48 Pac:











- ☐ Press **←** **LIBRARY** to display all libraries available to the HP 48SX. Find and press **SPICE** to enter the Spice48 Pac library directory. The screen displays new menukeys (softkeys) along the bottom, as shown:



Press **SPICE** (the first softkey) to start the application. To display a screen containing the revision number and product information about the Spice48 Pac, press **ABOUT** (the second softkey).

- ☐ Type **α** **α** **SPICE48** **ENTER** to start the application.
- ☐ Add the command SPICE48 to the CST (custom) menu. (For more information, refer to Chapter 15 of the HP 48SX Owner's Manual, "Customizing the Calculator.") After the command has been added to CST, press **CST** **SPICE** to start the application.


Moving Around the Screen

Use the  and  keys to move the pointer up and down in a menu screen. Press   to move the pointer to the bottom of the screen, or to page down one screen at a time if the pointer is already at the bottom of the screen. Press   to move the pointer to the top of the screen, or to page up one screen at a time. Press   to move the pointer to the very end of the menu or press   to move the pointer to the very beginning of the menu.

Using the Main Menu

After you start the application, the Main menu appears:



The Main menu lists the eight modules of the Spice48 Pac. A module is selected by moving the pointer to the desired item and pressing .

Items in the Main Menu





Each item in the Main menu is briefly described below and is discussed in detail in the various chapters of this manual.

Item	Description	See...
Circuit Manager	Adds, deletes, packs, and transfers circuit data; selects the current circuit; installs tutorial circuit data.	Chapter 3
Element Manager	Adds, deletes, and edits circuit elements.	Chapter 4
Analysis Manager	Performs calculations.	Chapter 5
Node Manager	Adds, deletes, and edits nodes; sets the reference node.	Chapter 6
Graphics Manager	Adds, deletes, and edits coordinates and creates the schematic.	Chapter 7

Data File Manager	Loads, deletes, and transfers stored data files.	Chapter 8
Miscellaneous Info	Reference data tables; resistor color chart calculations; standard component values; ASCII table.	Chapter 10

Summary of Operations

Screen	Softkeys
Main	ABOUT -STK PRINT VIEW FONT QUIT

Key	Action
ABOUT	Displays a screen containing the revision number and product information about the Spice48 Pac. Pressing any key erases the screen and returns to the previous menu or to the HP 48SX stack.
FONT	Toggles between the small and large fonts.
PRINT	Prompts for ONE or ALL to select items, and then sends those items to an IR printer.
QUIT	Quits the Spice48 Pac to the HP 48SX stack.
-STK	Prompts for ONE or ALL to select items, and then copies those items to the stack. The items are placed in a list if ALL was chosen.
VIEW	Displays the entire text of an item too wide to fit on the screen, up to one entire screen size. If the item fits on the screen, this key is non-functional.
	Quits the Spice48 Pac to the HP 48SX stack.
	Moves down one level in the menu structure, entering the module selected by the pointer.
 	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Changing the Font Size

The default font for the Spice48 Pac displays information in condensed, upper-case letters only. Pressing **FONT** will toggle the information to a larger font, which is case-sensitive:



The font size will remain the same until **FONT** is pressed again.

Viewing Items Too Wide for the Display

If the text of a menu item is too wide to fit within the display, an ellipsis (...) appears at the end of the line. On some screens, the **VIEW** softkey will be present—press **VIEW** to display the entire text of an item, up to one entire screen size. Once the full text has been displayed, press **ENTER** or **ATTN** to return to the menu. At all screens, including those screens where **VIEW** is not present, pressing **VIEW** will perform the same function. If an item does fit entirely on the screen, **VIEW** or **VIEW** will beep and do nothing.

Using the Search Mode

When menu lists are long, it is faster to locate an item using the search mode. To initiate a search, press **SEARCH** to display the following screen:



The HP 48SX is now locked in alpha-entry mode, as indicated by the alpha annunciator at the top of the screen. Alpha entry mode activates the white capital letters printed to the lower right of many keys. (For more information, refer below to “Alpha Lock” and to Chapter 2 of the HP 48SX Owner’s Manual, “The Keyboard and Display.”)

To perform a search, enter the first letter or letters of the desired string and press **ENTER**. The search function is case-sensitive, and will scan through all information in the current menu. To enter a lowercase letter in the alpha entry mode, precede the letter with **⏮**. To abort the search, press **ATTN**.

Text Editing

The softkeys present at the search screen and at many data input screens are command line editing keys. They allow you to edit the search string or input data. Their functions are summarized below. (For more information, refer to Chapter 3 of the HP 48SX Owner’s Manual, “The Stack and Command Line.”)






Summary of Operations

Screen	Softkeys
Text Editing	⏮SKIP SKIP⏭ ⏮DEL DEL⏭ INS ⏭STK










Key	Action
⏮DEL	Deletes all characters in the current word prior to the cursor.
DEL⏭	Deletes all characters in the current word between the cursor’s current position and the first character of the next word.
INS	Toggles between insert and type-over modes.
⏮SKIP	Moves the cursor to the beginning of the current word.
SKIP⏭	Moves the cursor to the beginning of the next word.
⏭STK	Activates the Interactive Stack, allowing arguments to be copied from the stack to the command line for editing by pressing ECHO .
ATTN	Clears the command line if there is text present, or aborts text entry if the command line is already blank.
ENTER	Accepts the current command line as the entry and returns to the previous menu or list.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”


Alpha Lock

System flag -60 controls whether or not Alpha Lock mode is set. The default setting for flag -60 is clear, which means that pressing  places the HP 48SX in alpha-entry mode for only one character, and you must press   to lock alpha-entry mode. If flag -60 is set, however, then pressing  only once locks alpha-entry mode. The examples in this manual assume that flag -60 is clear, so that each  keystroke turns on alpha-entry mode only for the following character. (For more information, refer to Chapter 2 of the HP 48SX Owner's Manual, "The Keyboard and Display.")

How to Load Data from the Stack

At all data input prompts, it is possible to copy values from the HP 48SX stack to the command line, even though the Spice48 Pac is executing. This is achieved through a limited version of the Interactive Stack. To activate the Interactive Stack at a data input prompt, press , or if that does not work, press   to display the EDIT menu and then press . At this point, unless the stack is empty, the screen will display the contents of the stack. Move the pointer up and down the stack by pressing  and , and when you reach the desired value, press  to copy it to the command line for editing. To exit the Interactive Stack and return to the command line, press  or . After returning to the command line, you can edit the value with the editing softkeys described above. (For more information, refer to Chapter 3 of the HP 48SX Owner's Manual, "The Stack and Command Line.")

System Flags

Executing the Spice48 Pac will not change the flag settings or stack depth on your HP 48SX, unless you push  at some point to leave results on the stack. For your convenience, most flag settings are preserved during operation of the Spice48 Pac, including the alpha-lock setting. However, for the software to operate properly, some system flags are temporarily modified during execution:

- ☐ Binary word size is set to 64
- ☐ Clock display is turned off
- ☐ Radix mark is set to "." (period)
- ☐ Last Arguments are saved
- ☐ User Mode is turned off

When you press **[ATTN]** or **[QUIT]** to exit the Spice48 Pac to the HP 48SX stack, all system and user flags are restored to their previous settings for your convenience.

WARNING: Pressing **[ATTN]** multiple times in rapid succession may abort the Spice48 Pac without resetting the state of your HP 48SX. Do not do this! The Pac is designed to be tolerant of a few **[ATTN]** presses, but it cannot properly restore your stack and flag settings if you push **[ATTN]** too many times in a row.

User Flags

The display font size is controlled by the setting of user flag 57. If flag 57 is clear, the smaller display font will be used; if flag 57 is set, the larger display font will be used. Changes in the display font during operation of the Spice48 Pac are preserved after you quit to the HP 48SX stack.

The state of units (on or off) is controlled by the setting of user flag 61. If flag 61 is clear, units are on; if flag 61 is set, units are off. Changes in the units status during operation of the Spice48 Pac are preserved after you quit to the HP 48SX stack. Note that units are only relevant in the Spice48 Pac when viewing data in the Constant Library in the Miscellaneous Info module. The units flag setting makes no difference to circuit calculations.

Memory Requirements

A minimum of about 1.7K free memory is required for this Pac to operate, but more free memory is required to analyze circuits and save data files. For example, a simple circuit of only a few nodes may require 1K to 5K to fully analyze, while a complex circuit of ten or more nodes may require 10K to 20K or more. The precise free memory requirements vary according to the exact size of your circuits, whether or not you create circuit schematics, and how many data points you create during each analysis.

In many cases, an estimate of the free memory required to complete an operation will be displayed. These estimates are generous, and if it appears that the amount of free memory required to complete an operation is only slightly more than the amount of free memory available, you should attempt the operation anyway, because it may succeed. Even if the operation fails and the Spice48 Pac quits due to insufficient memory, your circuit data will *not* be affected, although items may be left on the stack.

NOTE: Memory warnings are only provided during analysis and schematic preparations. You will not be warned of low memory when loading/decompressing a data file or when creating a plot.

If the Spice48 Pac appears to be functioning incorrectly—e.g., if you attempt to create a plot or view a data file and nothing happens—it is likely that there is not enough free memory in your HP-48SX to complete the operation. Possible solutions to the problem of too little free memory are:

- ☐ Decrease the number of data points for the desired analysis, until the memory required falls below the free memory. (For more information, refer to Chapter 5, “Analysis Manager.”)
- ☐ Pack unused circuits. (For more information, refer to “Packing the Current Circuit” in Chapter 3.)
- ☐ Clear all graphics information from circuits for which you have previously created schematics. (For more information, refer to “Clearing All Graphics” in Chapter 7.)
- ☐ Delete unwanted circuits and data files. You can first archive them to a computer if you wish. (For more information, refer to Chapter 3, “Circuit Manager,” and Chapter 9, “Data File Manager.”)
- ☐ Add additional free memory to your HP 48SX by merging a 32K or 128K RAM card. (For more information, refer to Chapter 5 of the HP 48SX Owner’s Manual, “Calculator Memory.”)

The 'SPARCOM' Directory

Most Sparcom Pocket Professional™ Pacs create a directory 'SPARCOM' in the HOME directory of your HP 48SX. Inside the 'SPARCOM' directory, each particular Pac creates a specific subdirectory—for the Spice48 Pac, the name of that directory is 'SPICE48D'. All circuits and data files for Spice48 are stored inside 'SPICE48D', so as not to conflict with your global variables and other directories. If you are extremely low on free memory and do not need to keep any of the Spice48 data files in your HP 48SX, you can purge the 'SPICE48D' directory (using the command PGDIR). The next time you execute the Spice48 Pac, the 'SPICE48D' directory will automatically be re-created, but your data files will be permanently lost unless they were uploaded to a computer for storage. (For more information, refer to Chapter 7 of the HP 48SX Owner’s Manual, “Directories.”)

Chapter 2

Spice48 Tutorial


The Spice48 Tutorial is designed to introduce you to the power and flexibility of Spice48 by stepping through the element entry, analysis, and schematic creation for two circuits: a simple RC circuit and a complex Bipolar Junction Transistor (BJT) DC Model.

This chapter covers:

- ☐ Using the Spice48 Tutorial
- ☐ Example: RC Circuit
- ☐ Example: Bipolar Junction Transistor, DC Model

Using the Spice48 Tutorial

To begin the Spice48 Tutorial, follow these steps:

- ❶ Press  **LIBRARY** to display all libraries available to your HP 48SX.
- ❷ Find and press **SPICE** to enter the Spice48 Pac library directory.
- ❸ Press the first softkey, **SPICE**, to start the Spice48 Pac.



You are now at the Main menu of Spice48.

Example: RC Circuit

This example is a simple RC circuit with a pulsed voltage source.

To investigate this circuit, You will follow these steps:

- ❶ Add the circuit RC.
- ❷ Add nodes to RC.
- ❸ Add elements to RC.
- ❹ Draw the schematic of RC.
- ❺ Analyze, plot, and save data about RC.

The following screens and free memory indicators assume you are executing Spice48 for the first time, from an empty HP 48SX without any merged RAM cards, that the display setting is standard (STD), and that your HP 48SX is in degrees mode (DEG). If your HP 48SX is configured otherwise, the screens may appear slightly different.

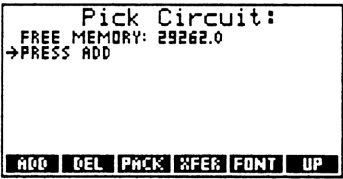
To complete this example requires about 6K free memory in your HP 48SX, although you can decrease that requirement if you do not save the data files as indicated or draw the schematic.

Adding the Circuit RC

First, a new circuit must be created. (For more information, refer to Chapter 3, “Circuit Manager.”)

At the Main menu, make sure the pointer is at “Circuit Manager” and press

ENTER:



Press **ADD** or **ENTER** to add a new circuit, and at the prompt type RC, the name of this circuit:



When you have finished, press **ENTER** to accept the name and return to the Circuit Manager. (For more information on using the editing softkeys, refer to “Text Editing” in Chapter 1.) A new subdirectory named RC will be created inside the 'SPICE48D' directory in your HP 48SX. It will be used to store information about the circuit RC, which is now listed:

```

Pick Circuit:
FREE MEMORY: 29051.5
→RC: 151.0 (PACKED) (CURRENT)

ADD DEL PACK XFER FONT UP

```

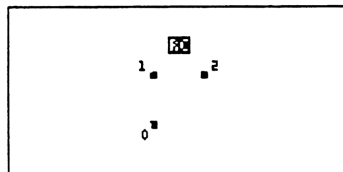
The “(Packed)” label indicates that the circuit RC contains no temporary calculation or graphics global variables. (For more information, refer to “Packing the Current Circuit” in Chapter 3.)

The “(Current)” label indicates that the circuit RC is the current circuit, meaning that all other operations of Spice48 will be performed on the circuit RC. (For more information, refer to “Selecting the Current Circuit” in Chapter 3.)

This completes Step ①. Press **UP** to return to the Main menu.

Adding Nodes to RC

Second, nodes must be added to the circuit. (For more information, refer to Chapter 6, “Node Manager.”) The following screen is an enhancement of one created by the Graphics Manager, with only nodes labeled:



This example will demonstrate how to add the nodes separately from the elements; the next example will demonstrate how to add nodes as necessary during element entry. (For more information, refer below to “Example: Bipolar Junction Transistor (BJT) DC Model.”)

At the Main menu, move the pointer to “Node Manager” and press **ENTER**:

Nodes:		
PRESS ADD		
ADD	DEL	EDIT REFER UP

Press **ADD** or **ENTER** to add new nodes, and at the prompt type 0 **SPC** 1 **SPC** 2, the names of the nodes in this circuit:

Nodes:		
PRG		
{ HOME SPARCOM SPICE48D RC }		
Enter node name(s):		
→ node1 node2 ...		
(< names)		
0 1 2		
←XIP XIP← DEL DEL→ INS ▢ TEST		

When you have finished, press **ENTER** to accept the names and return to the Node Manager. The new nodes are now listed:

Nodes:		
0	1	2
ADD	DEL	EDIT REFER UP

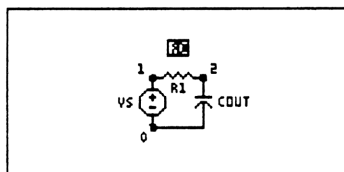
Press **▢** to move the cursor to node 0 and press **REFER** to set that as the reference (ground) node of circuit RC. Node 0 will be labeled with “(REFER)” in the Node Manager:

Nodes:		
(REFER) 0	1	2
ADD	DEL	EDIT REFER UP

This completes Step ②. Press **UP** to return to the Main menu.

Adding Elements to RC

Third, elements must be added to the circuit. (For more information, refer to Chapter 4, “Element Manager.”) The following screen is an enhancement of one created by the Graphics Manager, with both nodes and elements labeled:



You will follow these steps:

- ① Add the voltage source VS.
- ② Add the resistor R1.
- ③ Add the capacitor COUT.

At the Main menu, move the pointer to “Element Manager” and press **ENTER**:

Elements:			
PRESS ADD			
ADD	DEL	EDIT	UP

Adding Element VS

Press **ADD** or **ENTER** to add a new element, and at the prompt press **▶** **▼** to move the cursor to “V-Source”:

Pick element type:		
R-RESISTOR	C-CAPACITOR	L-INDUCTOR
K-COUPLED	V-SOURCE	I-SOURCE
E-VCVS	F-CCCS	G-VCCS
H-CCVS		
MAIN		UP

Press **ENTER** to accept the element type and display the element parameter screen for a voltage source:

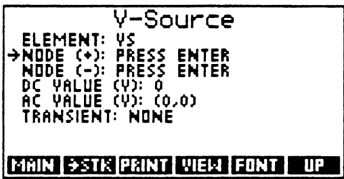
V-Source	
→ELEMENT:	PRESS ENTER
NODE (+):	PRESS ENTER
NODE (-):	PRESS ENTER
DC VALUE (V):	0
AC VALUE (V):	(0,0)
TRANSIENT:	NONE
MAIN	→SET PRINT VIEW FONT UP

Entering the Name of VS

Make sure the pointer is at “Element,” press **ENTER** to enter the name of this element, and at the prompt type S, to name this element VS (The element name must start with V.):

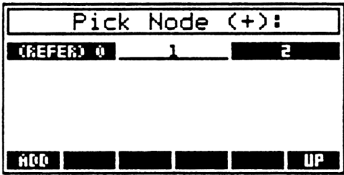


When you have finished, press **ENTER** to accept the element name and return to the element parameter screen. The pointer will automatically be moved down one line for your convenience:

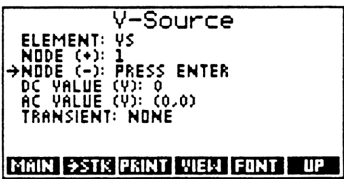


Entering the (+) Node of VS

Make sure the pointer is at “Node (+),” press **ENTER** to select the (+) node of this element, and at the prompt press **1**, to move the cursor to node 1:



Press **ENTER** to accept node 1 as the (+) node and return to the element parameter screen. The pointer will automatically be moved down one line for your convenience:



Entering the (-) Node of VS

Make sure the pointer is at “Node (-),” press **ENTER** to select the (-) node of this element, and at the prompt the cursor will be on node 0. Press **ENTER** to accept node 0 as the (-) node and return to the element parameter screen. The pointer will automatically be moved down one line for your convenience:

```

V-Source
ELEMENT: VS
NODE (+): 1
NODE (-): 0
→DC VALUE (V): 0
AC VALUE (V): (0,0)
TRANSIENT: NONE
MAIN →STE PRINT VIEW FONT UP
```

Entering the DC Value of VS

Make sure the pointer is at “DC Value (V),” press **ENTER** to enter the DC value of this element, and at the prompt press **ATTN** or **←** to clear the command line and type 1, to set the DC value to 1 V. Press **ENTER** to accept the value and return to the element parameter screen. The pointer will automatically be moved down one line for your convenience:

```

V-Source
ELEMENT: VS
NODE (+): 1
NODE (-): 0
DC VALUE (V): 1
→AC VALUE (V): (0,0)
TRANSIENT: NONE
MAIN →STE PRINT VIEW FONT UP
```

Entering the AC Value of VS

Make sure the pointer is at “AC Value (V),” press **ENTER** to enter the AC value of this element, and at the prompt press **←←←←DEL** 1 or **ATTN←÷** 1 **←.** 0, to set the AC value to (1,0):

```

PRG
{ HOME SPARCOM SPICE48D RC }
Edit AC Value (V):
→ cmplx (m,∠) or (r,i)
(1∠0)
←SKIP←SKIP←DEL DEL→INS→STE
```

When you have finished, press **ENTER** to accept the value and return to the element parameter screen. The pointer will automatically be moved down one line for your convenience:

```

V-Source
ELEMENT: VS
NODE (+): 1
NODE (-): 0
DC VALUE (V): 1
AC VALUE (V): (1,0)
→TRANSIENT: NONE
MAIN  →STK PRINT VIEW FONT  UP

```

Entering the Transient of VS

Make sure the pointer is at “Transient,” press **ENTER** to enter the transient of this element, and at the prompt press **▶**, to select PULSE as the transient type. Press **ENTER** to accept the transient type and display the transient parameter screen for a pulse transient:

```

Pulse Transient
→INITIAL VALUE (V,A): PRESS ENTER
PULSED VALUE (V,A): PRESS ENTER
DELAY TIME (S): PRESS ENTER
RISE TIME (S): PRESS ENTER
FALL TIME (S): PRESS ENTER
PULSE WIDTH (S): PRESS ENTER
PERIOD (S): 9.E499
MAIN  →STK PRINT VIEW FONT  UP

```

The desired pulse will jump to 10 V from an initial value of 0 V after a delay of 0.1 sec, with a pulse width of 1 sec, rise and fall times of 1 μ sec, and a period of 100 sec. (For more information, refer to “Pulse Transient” in Chapter 4.) Enter these values in the appropriate parameters in the same manner as before, until your screen appears as follows (To quickly enter values in μ sec, press **MICRO** after the value at the prompt.):

```

Pulse Transient
→INITIAL VALUE (V,A): 0
PULSED VALUE (V,A): 10
DELAY TIME (S): .1
RISE TIME (S): .000001
FALL TIME (S): .000001
PULSE WIDTH (S): 1
PERIOD (S): 100
MAIN  →STK PRINT VIEW FONT  UP

```

When you have finished, press **UP** to return to the element parameter screen, and the transient will be converted to a user-defined function and saved automatically. The element parameter screen appears:

```

V-Source
→ELEMENT: VS
NODE (+): 1
NODE (-): 0
DC VALUE (V): 1
AC VALUE (V): (1,0)
TRANSIENT: PULSE
MAIN  →STK PRINT VIEW FONT  UP

```

This completes Step ③①. Press **UP** to return to the Element Manager:

Elements:	
VS	
ADD	DEL EDIT UP

Because this is the first element of circuit RC and the reference node remains unspecified, the (-) node (node 0) of this element will automatically be set as the reference node.

Adding Element R1

Press **ADD** to add a new element, and at the prompt the cursor appears on “R-Resistor”:

Pick element type:		
R-RESISTOR	C-CAPACITOR	L-INDUCTOR
K-COUPLED	V-SOURCE	I-SOURCE
E-VCVS	F-CCCS	G-WCCS
H-CCVS		
MAIN		UP

Press **ENTER** to accept the element type and display the element parameter screen for a resistor:

R-Resistor	
→ELEMENT:	PRESS ENTER
NODE (+):	PRESS ENTER
NODE (-):	PRESS ENTER
VALUE (Ω):	PRESS ENTER
MAIN	←STR PRINT VIEW FONT UP

Entering the Name of R1

Make sure the pointer is at “Element,” press **ENTER** to enter the name of this element, and at the prompt type 1, to name this element R1 (The element name must start with R.). When you have finished, press **ENTER** to accept the element name and return to the element parameter screen:

R-Resistor	
ELEMENT:	R1
→NODE (+):	PRESS ENTER
NODE (-):	PRESS ENTER
VALUE (Ω):	PRESS ENTER
MAIN	←STR PRINT VIEW FONT UP

Entering the (+) Node of R1

Make sure the pointer is at “Node (+),” press **ENTER** to select the (+) node of this element, and at the prompt press **▶**, to move the cursor to node 1. Press **ENTER** to accept node 1 as the (+) node and return to the element parameter screen:

```

R-Resistor
ELEMENT: R1
NODE (+): 1
→NODE (-): PRESS ENTER
VALUE (Ω): PRESS ENTER

MAIN ▶STE PRINT VIEW FONT UP

```

Entering the (–) Node of R1

Make sure the pointer is at “Node (–),” press **ENTER** to select the (–) node of this element, and at the prompt press **◀**, to move the cursor to node 2. Press **ENTER** to accept node 2 as the (–) node and return to the element parameter screen:

```

R-Resistor
ELEMENT: R1
NODE (+): 1
NODE (-): 2
→VALUE (Ω): PRESS ENTER

MAIN ▶STE PRINT VIEW FONT UP

```

Entering the Value of R1

Make sure the pointer is at “Value (Ω),” press **ENTER** to enter the value of this element, and at the prompt type 1, to set the value to 1Ω. Press **ENTER** to accept the value and return to the element parameter screen:

```

R-Resistor
→ELEMENT: R1
NODE (+): 1
NODE (-): 2
VALUE (Ω): 1

MAIN ▶STE PRINT VIEW FONT UP

```

This completes Step ③②. Press **UP** to return to the Element Manager:

```

Elements:
VS      R1

ADD DEL EDIT      UP

```

Adding Element COUT

Press **ADD** to add a new element, and at the prompt press **▶** to move the cursor to “C-Capacitor”. Press **ENTER** to accept the element type and display the element parameter screen for a capacitor:

```

C-Capacitor
→ELEMENT: PRESS ENTER
NODE (+): PRESS ENTER
NODE (-): PRESS ENTER
VALUE (F): PRESS ENTER
INITIAL (V): 0
MAIN  ⤴STK PRINT VIEW FONT  UP

```

Entering the Name of COUT

Make sure the pointer is at “Element,” press **ENTER** to enter the name of this element, and at the prompt type OUT, to name this element COUT (The element name must start with C.). When you have finished, press **ENTER** to accept the element name and return to the element parameter screen.

Entering the (+) Node of COUT

Make sure the pointer is at “Node (+),” press **ENTER** to select the (+) node of this element, and at the prompt press **◀**, to move the cursor to node 2. Press **ENTER** to accept node 2 as the (+) node and return to the element parameter screen.

Entering the (–) Node of COUT

Make sure the pointer is at “Node (–),” press **ENTER** to select the (–) node of this element, and at the prompt the cursor will be on node 0. Press **ENTER** to accept node 0 as the (–) node and return to the element parameter screen.

Entering the Value of COUT

Make sure the pointer is at “Value (F),” press **ENTER** to enter the value of this element, and at the prompt type 1, to set the value to 1 F. Press **ENTER** to accept the value and return to the element parameter screen.

Entering the Initial Value of COUT

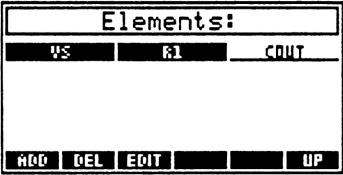
Make sure the pointer is at “Initial (V),” press **ENTER** to enter the initial value of this element, and at the prompt type **ATTN** 5 **⌘**, to set the value to –5 V. Press **ENTER** to accept the value and return to the element parameter screen:

```

C-Capacitor
→ELEMENT: COUT
NODE (+): 2
NODE (-): 0
VALUE (F): 1
INITIAL (V): -5
MAIN  ⤴STK PRINT VIEW FONT  UP

```

This completes Step ③③. Press **UP** to return to the Element Manager:



This completes all of Step ③. Press **UP** to return to the Main menu.

Drawing the Schematic of RC

Fourth, the circuit schematic can be drawn. (For more information, refer to Chapter 7, “Graphics Manager.”) This step can occur at any point after the circuit has been fully specified (steps ① through ③).

You will follow these steps:

- ① Set the node coordinates.
- ② Automatically generate the element coordinates and orientations.
- ③ Automatically generate the wires.
- ④ View the schematic.

(For more information about coordinates, refer to “Creating an Accurate Schematic” in Chapter 7.)

At the Main menu, move the pointer to “Graphics Manager” and press **ENTER**:



Setting Node Coordinates

Make sure the pointer is at “Set Node Coordinates” and press **ENTER**. The circuit nodes will be displayed:

Nodes:		
0	1	2
MAIN	EDIT	CLEAR UP

Make sure the pointer is at node 0, press **EDIT** or **ENTER** to enter set the coordinates, and at the prompt type 0 **↵** 0, to set the coordinates to (0,0):

PRG	
{ HOME SPARCOM SPICE4BD RC }	
Edit coordinates:	
→ x y	
< integers >	
0 0	
←SKIP←SKIP←DEL DEL→INS→↑STX	

When you are finished, press **ENTER** to accept the coordinates and return to the node screen:

Nodes:		
(0.0) 0	1	2
MAIN	EDIT	CLEAR UP

In a similar manner, set the coordinates of node 1 to (0,1) and of node 2 to (1,1), until your screen appears as follows:

Nodes:		
(0.0) 0	(0.1) 1	(1.1) 2
MAIN	EDIT	CLEAR UP

This completes Step 4①. Press **UP** to return to the Graphics Manager.

Setting Element Coordinates and Orientations

Move the pointer to “Set Elmt Coordinates” and press **ENTER**. The circuit elements will be displayed:

Elements:		
VS	R1	COUT
MAIN	EDIT	AUTO CLEAR UP

Press **AUTO** to automatically generate the element coordinates and orientations from the node coordinates. The elements should be located as follows:

Elements:		
(0,1),1	VS	(0,1),2 R1 (1,1),1 COUT
MAIN	EDIT	AUTO CLEAR UP

This completes Step 4②. Press **UP** to return to the Graphics Manager.

Setting Wire Coordinates

Move the pointer to “Set Wire Coordinates” and press **ENTER**. No wires currently exist:

Wires:		
PRESS ADD		
ADD	DEL	EDIT AUTO CLEAR UP

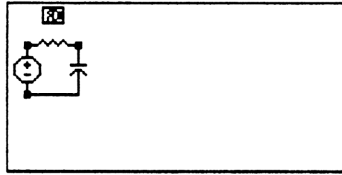
Press **AUTO** to automatically generate the wires necessary to fix “loose” connections in the circuit. Only one wire is necessary to complete the circuit RC schematic, and it should be located as follows:

Wires:		
(0,0)	(1,0)	
ADD	DEL	EDIT AUTO CLEAR UP

This completes Step 4③. Press **UP** to return to the Graphics Manager.

Viewing the Schematic

Move the pointer to “View Schematic” and press **ENTER**:



When you have finished viewing the schematic, press **ENTER** to return to the Graphics Manager.

This completes Step ④④ and therefore all of Step ④. Press **UP** to return to the Main menu.

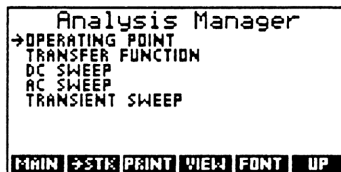
Analyzing RC

Fifth, the circuit can be analyzed and the resulting data viewed, plotted, and saved. (For more information, refer to Chapter 5, “Analysis Manager” and Chapter 9, “Output Manager.”). This step can occur at any point after the circuit has been fully specified (steps ① through ③).

You will perform the following analyses:

- ① Operating Point (OP) analysis to find the steady-state voltages and currents at the corner frequency $1/(2\pi RC) = 0.159$.
- ② Transfer Function (TF) analysis to determine the input impedance, output impedance, and gain at the corner frequency $1/(2\pi RC) = 0.159$.
- ③ AC Sweep (AC) analysis to create Bode plots of dB vs. frequency and phase vs. frequency. Sweep frequency from 0.0159 Hz to 1.59 Hz.
- ④ Transient Sweep (TR) analysis to determine the transient response of circuit RC to the input pulse on the voltage source VS.

At the Main menu, move the pointer to “Analysis Manager” and press **ENTER**:

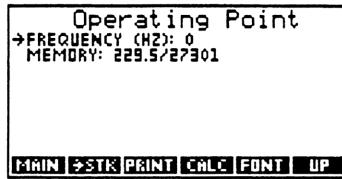


Operating Point (OP) Analysis

Analysis: Determine the steady-state operation point (or the “Q-point”) of circuit RC at the corner frequency $1/(2\pi RC) = 0.159$.

Performing an OP Analysis

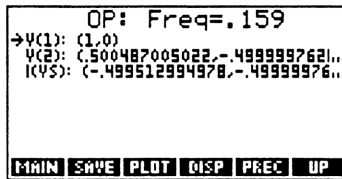
Make sure the pointer is at “Operating Point” and press **ENTER**. There will be a delay (~ 30 sec) while the circuit is prepared, and then the analysis parameter screen appears:



Change the frequency to 0.159 Hz and press **CALC**. There will be a delay (~ 7 sec) while the analysis is performed.

Viewing the Results (Data View)

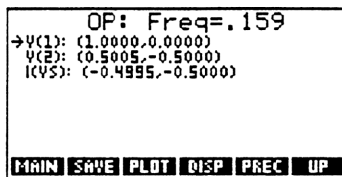
When the calculation completes, the OP results will be displayed in the Output Manager Data View, which (in this case) displays voltage and current data (For more information, refer to “Data View” in Chapter 9.):



The output nodes for the OP analysis include V(1), the voltage at node 1, V(2), the voltage at node 2, and I(VS), the current through the voltage source VS.

Changing Display Precision

Press **PREC** **▶** **ENTER** 4 **ENTER** to change the display precision to four decimal places:



(For more information, refer to “Display Forms” in Chapter 9.)

Changing Display Type

Press **DISP** **▶** **▼** **ENTER** to change the display type to phase angle:

```

OP: Freq=.159
→V(1): 0.0000
V(2): -44.9721
I(VS): -134.9721

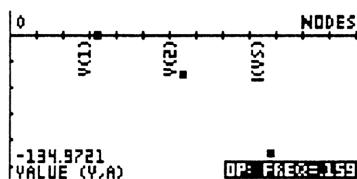
MAIN SAVE PLOT DISP PREC UP

```

(For more information, refer to “Display Forms” in Chapter 9.)

Plotting Node Values (Node Plot)

Press **PLOT** to plot the OP results, and at the prompt, press **ENTER** to accept the default vertical range. The following plot appears:



This node plot displays the phase angle of each output node. Points in node plots are never connected because they represent distinct output nodes. (For more information, refer to “Node Plots” in Chapter 9.)

When you have finished viewing the plot, press **ATTN** to return to the Output Manager Data View.

Checking Free Memory

Press **NXT** **MEM** to determine the free memory required to save the OP analysis data:

```

Size if saved:
146.5 bytes

Free memory:
25533.5 bytes

PRESS [ENTER] TO RETURN TO LIST ...

```

Press **ENTER** or **ATTN** to return to the Output Manager Data View.

Saving the Results

If you have enough free memory, press **NXT** **SAVE**, and at the prompt press **ENTER** to accept the default name. The OP analysis data will be stored into the file 'RC.OP' in the 'fileS' subdirectory of the 'SPICE48D' directory. (For more information, refer to Chapter 8, “Data File Manager.”)

This completes Step 5①. Press **UP** to return to the Analysis Manager.

Transfer Function (TF) Analysis

Analysis: Determine the input impedance, output impedance, and gain of circuit RC at the corner frequency $1/(2\pi RC) = 0.159$.

Performing a TF Analysis

Make sure the pointer is at “Transfer Function” and press **ENTER**. The analysis parameter screen appears:

```
Transfer Function
→SOURCE: PRESS ENTER
OUTPUT NODE (+): PRESS ENTER
OUTPUT NODE (-): 0
FREQUENCY (HZ): 0.0000
MEMORY: 180/25947

MAIN →STR PRINT CALC FONT UP
```

Select VS as the source, select node 2 as the output node (+), change the frequency to 0.159 Hz, and press **CALC**. There will be a delay (~ 18 sec) while the analysis is performed.

Viewing the Results (Data View)

When the calculation completes, the TF results will be displayed in the Output Manager Data View, which (in this case) displays a variety of values:

```
TF: Results
→SOURCE: VS
OUTPUT NODE (+): 2
OUTPUT NODE (-): 0
FREQUENCY (HZ): 0.1590
GAIN: (0.5005,-0.5000)
INPUT IMPEDANCE (Ω): (1.0000,-1.1...
OUTPUT IMPEDANCE (Ω): (0.5005,-1.1...

MAIN SAVE PLOT DISP PREC UP
```

The TF results consist of the source, the output (+) and (–) nodes, the frequency, the gain, and the input and output impedances, where the input impedance is measured across the source and the output impedance is measured between the output (+) and (–) nodes. The impedances and the gain are all complex numbers because of the non-zero frequency.

This completes Step 5②. Press **UP** to return to the Analysis Manager.

AC Sweep (AC) Analysis

Analysis: Create Bode plots of dB vs. frequency and phase vs. frequency for circuit RC, sweeping the frequency from 0.0159 Hz to 1.59 Hz.

Performing an AC Analysis

Make sure the pointer is at “AC Sweep” and press **ENTER**. The analysis parameter screen appears:

```
AC Sweep
→START (HZ): PRESS ENTER
STOP (HZ): PRESS ENTER
NO. POINTS: PRESS ENTER
OUTPUT SCALE: DECADE
MEMORY: ???/25803.5

MAIN →STE PRINT CALC FONT UP
```

Set the start frequency to 0.0159 Hz, set the stop frequency to 1.59 Hz, set the number of points to 4 per decade, and press **CALC**. There will be a delay (~ 30 sec) while the analysis is performed.

Viewing the Results (Node View)

When the calculation completes, the AC results will be displayed in the Output Manager Node View, which displays all output nodes (For more information, refer to “Node View” in Chapter 9.):

```
AC: Nodes
-----
V(1)  V(2)  I(VS)
-----
MAIN MARK PLOT NODE= FREQ UP
```

The output nodes for the AC analysis include V(1), the voltage at node 1, V(2), the voltage at node 2, and I(VS), the current through the voltage source VS, for a range (sweep) of frequencies from 0.0159 Hz to 1.59 Hz.

Viewing the Results (Data View)

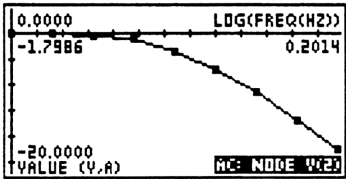
Examine output node V(2), the voltage at node 2, for each frequency. Press **▶** to move the cursor to V(2) and press **ENTER** to switch to the Output Manager Data View, which (in this case) displays voltage data (For more information, refer to “Data View” in Chapter 9.):

```
AC: Node V(2)
▼
→0.0159: -0.0431
0.0283: -0.1350
0.0503: -0.4132
0.0894: -1.1813
0.1590: -3.0061
0.2827: -6.1869
0.5028: -10.4062
0.8941: -15.1270
MAIN SAVE PLOT DISP PREC UP
```

For AC analysis data, the default display type is always dB.

Plotting dB vs. Frequency (Value Plot, Line)

Press **PLOT** to create a dB vs. frequency Bode plot, then **ENTER** to select a line plot, then **ENTER** to accept the default horizontal range, then **ATTN** 20 **+/-** **SPC** 0 **ENTER** to adjust the vertical range slightly. The following plot appears:



This value plot displays the voltage of output node V(2) as a function of frequency. Points in value plots are connected because they represent the same output node. The horizontal range for an AC analysis with a DECADE (or OCTAVE) output scale is always plotted logarithmically. (For more information, refer to “Value Plots” in Chapter 9.)

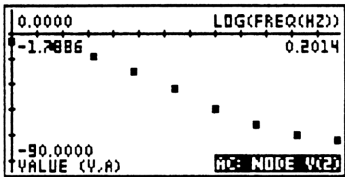
When you have finished viewing the plot, press **ATTN** to return to the Output Manager Data View, and press **UP** to return to the Output Manager Node View.

Changing Display Type

Press **NXT** **DISP** **ENTER** to change the display type to phase angle. (For more information, refer to “Display Forms” in Chapter 9.)

Plotting Phase vs. Frequency (Value Plot, Point)

Press **NXT** **PLOT** to create a phase vs. frequency Bode plot, then **ENTER** to select a point plot, then **ENTER** to accept the default horizontal range, then **ATTN** 90 **+/-** **SPC** 0 **ENTER** to adjust the vertical range slightly. The following plot appears:



This is also a semi-logarithmic value plot of output node V(2), as described above.

When you have finished viewing the plot, press **ATTN** to return to the Output Manager Node View.

Saving the Results

Press **NXT** **MEM** to determine the free memory required to save the AC analysis data. If you have enough free memory, press **SAVE**, and at the prompt press **ENTER** to accept the default name. The AC analysis data will be stored into the file 'RC.AC' in the 'files' subdirectory of the 'SPICE48D' directory. (For more information, refer to Chapter 8, "Data File Manager.")

This completes Step 5③. Press **UP** to return to the Analysis Manager.

Transient Sweep (TR) Analysis

Analysis: Determine the transient response of circuit RC to the input pulse on the voltage source VS. Examine the charge on the capacitor COUT and the current flow through the voltage source VS as a function of time.

Performing a TR Analysis

Make sure the pointer is at "Transient Sweep" and press **ENTER**. The analysis parameter screen appears:

```
Transient Sweep
→SAVE START (S): PRESS ENTER
SAVE STOP (S): PRESS ENTER
SAVE STEP (S): 0.1250
CALC. STEP (S): 0.1250
USE OP OR IC: IC
CALC. SPEED: FAST
MEMORY: ???/25277
MAIN ←STK PRINT CALC FONT UP
```

Set the save start to 0 sec, set the save stop to 4 sec, set the save step to 0.2 sec, change the calc step to 0.2 sec, and press **CALC**. There will be a delay (~ 40 sec) while the analysis is performed.

Viewing the Results (Node View)

When the calculation completes, the TR results will be displayed in the Output Manager Node View, which displays all output nodes (For more information, refer to "Node View" in Chapter 9.):

```
TR: Nodes
V(1)  V(2)  I(VS)
MAIN MARK PLOT NODE TIME UP
```

The output nodes for the TR analysis include V(1), the voltage at node 1, V(2), the voltage at node 2, and I(VS), the current through the voltage source VS, for a range (sweep) of times from 0 sec to 4 sec every 0.2 sec.

Viewing the Results (Time View)

Press **TIME** to switch to the Output Manager Time View, which displays all times at which TR analysis data was saved (For more information, refer to “Time View” in Chapter 9.):

TR: Times		
0.0000	0.2000	0.4000
0.6000	0.8000	1.0000
1.2000	1.4000	1.6000
1.8000	2.0000	2.2000
2.4000	2.6000	2.8000
MAIN MARK PLOT NODE TIME UP		

Viewing the Results (Data View)

Examine the output node voltages and currents at time 2.000. Press **▶▼▼▼** to move the cursor to 2.000 and press **ENTER** to switch to the Output Manager Data View, which (in this case) displays voltage and current data (For more information, refer to “Data View” in Chapter 9.):

TR: Time=2.0000	
→V(1):	0.0000
V(2):	2.1593
I(V5):	2.1593
MAIN SAVE PLOT DISP PREC UP	

When you have finished viewing the data, press **UP** to return to the Output Manager Time View and press **NODE** to switch to the Output Manager Node View.

Changing Display Precision

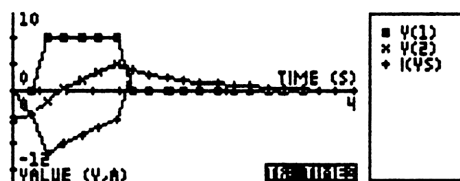
Press **NXT PREC ◀ ENTER** to change the display precision to standard. (For more information, refer to “Display Forms” in Chapter 9.)

Marking All Data

Press **NXT MARK MARK MARK** to tag all three output nodes with triangular marks.

Plotting Voltage/Current vs. Time (Overlay Value Plot, Lines)

Press **PLOT** to create an overlay plot, then **ENTER** to select a line plot, then **ENTER** to accept the default horizontal range, then **ATTN 12 +/- SPC 10 ENTER** to adjust the vertical range slightly. The following plot (and symbol key) appears:



Press **◀** to scroll the plot to the left, **▶** to scroll the plot to the right, or **◀▶** to jump to the far left edge of the plot, **▶▶** to jump to the far right edge of the plot. Press **GRAPH** to toggle between the scrolling mode and the HP 48SX Graphics Environment. (For more information, refer to “Value Plots” in Chapter 9 of this manual and to Chapter 18 of the HP 48SX Owner’s Manual, “Basic Plotting and Function Analysis.”)

This value plot displays the voltages for nodes 1 and 2 and the current through VS as a function of time. The ■ symbols represent the pulse transient on the voltage source. The × symbols represent the capacitor voltage, which starts initially at -5 V, charges up to a maximum of about 5 V, and then decays towards 0 V. The + symbols represent the current flow through the voltage source, which changes sharply in accordance with the pulse transient. (For more information, refer to Chapter 9, “Output Manager.”)

When you have finished viewing the plot, press **ATTN** to return to the Output Manager Node View.

Saving the Results

Press **NXT MEM** to determine the free memory required to save the TR analysis data. If you have enough free memory, press **SAVE**, and at the prompt press **ENTER** to accept the default name. The TR analysis data will be stored into the file 'RC.TR' in the 'fileS' subdirectory of the 'SPICE48D' directory. (For more information, refer to Chapter 8, “Data File Manager.”)

This completes Step ⑤④. Press **UP** to return to the Analysis Manager.

This completes all of Step ⑤. Press **UP** to return to the Main menu.

This completes all of Example: RC Circuit.

Example: Bipolar Junction Transistor, DC Model

The second example is a DC BJT circuit model. Strictly speaking, BJT transistors are modeled by a complicated, non-linear relation, but since Spice48 is only a linear circuit analyzer, simpler linear models must be used.

The DC model is usually sufficient for BJTs in the active region, i.e., when the base-emitter junction is forward-biased and the collector-base junction reverse-biased. The model described in this example is of an NPN BJT, but by reversing the polarity of source VBE (and thus the direction of current flow at the base and collector), it could be easily modified to that of a PNP BJT.

In the base circuit, the source VBE accounts for the base-emitter DC voltage drop. (The voltage of source VBE is actually output node V(VBE) for the analyses below.) V(VBE) is not always exactly known, and different rules of thumb suggest values ranging from 0.60 V to 0.75 V. Another component of the model is the current-controlled current source FCE in the collector-emitter circuit.

Source FCE is governed by the DC current gain, β_{DC} , according to

$\beta_{DC} = I_C / I_B$, where I_C is the collector current and I_B the base current. (The base current is actually output node I(VBE) in the analyses below, because the base current is measured through source VBE.) β_{DC} is typically on the order of 100 or more.

One problem with Spice48 arises whenever the base current becomes reverse-biased. In this case, the transistor is cut-off, and no collector current should flow. However, because Spice48 is linear only, it quite readily calculates negative values for I(VBE). You must observe and note that these negative values signify cut-off, an invalid operating condition for actual transistors.

A second problem with Spice48 occurs whenever the base current becomes so great that the collector current causes physically impossible voltage drops in the external resistors; this forward-biases the collector-base junction and saturates the transistor. This situation is most readily indicated by a value of the collector-emitter voltage that has an opposite polarity to that during normal operation. You must detect this as a negative collector-emitter voltage for NPN BJTs or as a positive collector-emitter voltage for PNP BJTs.

To summarize, normal operation requires $I(VBE) > 0$ and either $V_{CE} > 0$ (NPN) or $V_{CE} < 0$ (PNP). (V_{CE} is actually $V(\text{COLLECTOR}) - V(\text{EMITTER})$ in the analyses below.)

To investigate this circuit, you will follow these steps:

- ❶ Add the circuit BJTDC.
- ❷ Add elements and nodes to BJTDC.
- ❸ Draw the schematic of BJTDC.
- ❹ Analyze, plot, and save data about BJTDC.

The following screens and free memory indicators assume you have just completed Example: RC Circuit.

To complete this example requires about 6K free memory in your HP 48SX, although you can decrease that requirement if you do not save the data files as indicated or draw the schematic.

Adding the Circuit BJTDC

First, a new circuit must be created.

At the Main menu, make sure the pointer is at “Circuit Manager” and press **ENTER**:

```
Pick Circuit:
FREE MEMORY: 24928.0
→RC: 2244.5 (CURRENT)

ADD DEL PACK XFER FONT UP
```

Press **ADD** to add a new circuit, and at the prompt type BJTDC, the name of this circuit. When you have finished, press **ENTER** to accept the name and return to the Circuit Manager:

```
Pick Circuit:
FREE MEMORY: 25194.5
→BJTDC: 151.0 (PACKED) (CURRENT)
RC: 2244.5

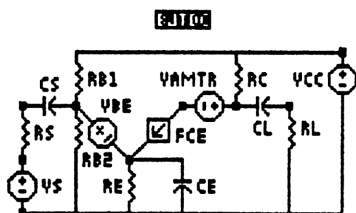
ADD DEL PACK XFER FONT UP
```

The “(Packed)” label indicates that the circuit BJTDC contains no temporary calculation or graphics global variables. The “(Current)” label indicates that the circuit BJTDC is the current circuit, meaning that all other operations of Spice48 will be performed on the circuit BJTDC.

This completes Step ❶. Press **UP** to return to the Main menu.

Adding Elements and Nodes to BJTDC

Second, elements and nodes must be added to the circuit. The following screen is an enhancement of one created by the Graphics Manager, with elements labeled:

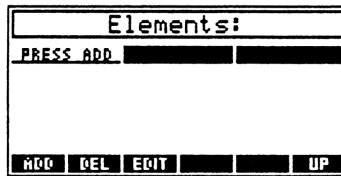


This example will demonstrate how to add nodes as necessary during element entry; the previous example demonstrated how to add the nodes separately from the elements. (For more information, refer above to “Example: RC Circuit.”)

You will follow these steps:

- ① Add the voltage source VS and the nodes SOURCE and GROUND.
- ② Add the resistor RS and the node INPUT.
- ③ Add the capacitor CS and the node BASE.
- ④ Add the resistors RB1 and RB2 and the node BATTERY.
- ⑤ Add the voltage source VBE, the resistor RE, the capacitor CE, and the node EMITTER.
- ⑥ Add the current-controlled current source FCE and the node AMMETER. The current-controlled current source should actually connect to the node COLLECTOR, but an ammeter will be inserted between nodes AMMETER and COLLECTOR to measure the current at the collector.
- ⑦ Add the voltage source VAMTR and the node COLLECTOR. VAMTR, the ammeter node, is just a 0 V source.
- ⑧ Add the resistor RC.
- ⑨ Add the resistor RL and the capacitor CL and the node OUTPUT.
- ⑩ Add the voltage source VBATT.

At the Main menu, move the pointer to “Element Manager” and press **ENTER**:



Adding VS and Nodes SOURCE and GROUND

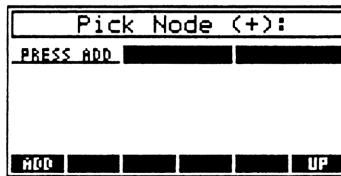
Press **ADD** or **ENTER** to add a new element, move the cursor to “V–Source”, and press **ENTER** to display the element parameter screen for a voltage source.

Entering the Name of VS

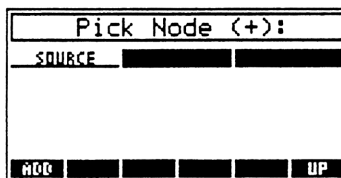
Make sure the pointer is at “Element,” press **ENTER** to enter the name of this element, and at the prompt type S, to name this element VS. (The element name must start with V.) When you have finished, press **ENTER** to accept the element name and return to the element parameter screen.

Entering the (+) Node of VS

Make sure the pointer is at “Node (+)” and press **ENTER** to select the (+) node of this element, although it does not yet exist in BJTDC:



Press **ADD** or **ENTER** to add a new node, and at the prompt type SOURCE, the name of the (+) node for VS. When you have finished, press **ENTER** to accept the name and return to the Node Manager. The new node is now listed:



Press **ENTER** to accept node SOURCE as the (+) node and return to the element parameter screen.

Entering the (-) Node of VS

Make sure the pointer is at “Node (-)” and press **ENTER** to select the (-) node of this element, although it does not yet exist in circuit BJTDC. Press **ADD** to add a new node, and at the prompt type GROUND, the name of the (-) node for VS. When you have finished, press **ENTER** to accept the name and return to the Node Manager. The new node is now listed. Press **ENTER** to accept node GROUND as the (-) node and return to the element parameter screen.

Entering the DC Value of VS

Do not change the default value of 0 V.

Entering the AC Value of VS

Change the value to (0.1,0).

Entering the Transient of VS

Do not change the default value of NONE.

This completes Step ②①. Press **UP** to return to the Element Manager. Because this is the first element of circuit BJTDC and the reference node remains unspecified, the (-) node (GROUND) of this element will automatically be set as the reference node.

Adding RS and Node INPUT

Press **ADD** to add a new element, leave the cursor at “R–Resistor”, and press **ENTER** to display the element parameter screen for a resistor. Enter element parameters until your screen appears as follows, making sure to add the node INPUT when selecting the (-) node of RS:



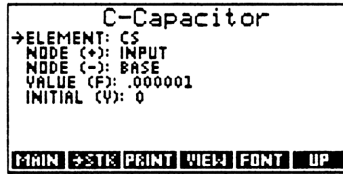
```
R-Resistor
→ELEMENT: RS
NODE (+): SOURCE
NODE (-): INPUT
VALUE (Ω): 1000

MAIN <= STK PRINT VIEW FONT UP
```

This completes Step ②②. Press **UP** to return to the Element Manager.

Adding CS and Node BASE

Press **ADD** to add a new element, move the cursor to “C–Capacitor”, and press **ENTER** to display the element parameter screen for a capacitor. Enter element parameters until your screen appears as follows, making sure to add the node BASE when selecting the (-) node of CS:



This completes Step ②③. Press **UP** to return to the Element Manager.

Adding RB1, RB2 and Node BATTERY

Press **ADD** to add a new element, leave the cursor at “R–Resistor”, and press **ENTER** to display the element parameter screen for a resistor. Enter element parameters until your screen appears as follows, making sure to add the node BATTERY when selecting the (+) node of RB1:



Press **UP** to return to the Element Manager.

Press **ADD** to add a new element, leave the cursor at “R–Resistor”, and press **ENTER** to display the element parameter screen for a resistor. Enter element parameters until your screen appears as follows:



This completes Step ②④. Press **UP** to return to the Element Manager.

Adding VBE, RE, CE, and Node EMITTER

Press **ADD** to add a new element, move the cursor to “V–Source”, and press **ENTER** to display the element parameter screen for a voltage source. Enter element parameters until your screen appears as follows, making sure to add the node EMITTER when selecting the (–) node of VBE:

```

V-Source
→ELEMENT: VBE
NODE (+): BASE
NODE (-): EMITTER
DC VALUE (V): .65
AC VALUE (V): (0,0)
TRANSIENT: NONE
MAIN ←STK PRINT VIEW FONT UP

```

Press **UP** to return to the Element Manager.

Press **ADD** to add a new element, leave the cursor at “R-Resistor”, and press **ENTER** to display the element parameter screen for a resistor. Enter element parameters until your screen appears as follows:

```

R-Resistor
→ELEMENT: RE
NODE (+): EMITTER
NODE (-): GROUND
VALUE (Ω): 5000
MAIN ←STK PRINT VIEW FONT UP

```

Press **UP** to return to the Element Manager.

Press **ADD** to add a new element, move the cursor to “C-Capacitor”, and press **ENTER** to display the element parameter screen for a capacitor. Enter element parameters until your screen appears as follows:

```

C-Capacitor
→ELEMENT: CE
NODE (+): EMITTER
NODE (-): GROUND
VALUE (F): .0001
INITIAL (V): 0
MAIN ←STK PRINT VIEW FONT UP

```

This completes Step 2⑥. Press **UP** to return to the Element Manager.

Adding FCE and Node AMMETER

Press **ADD** to add a new element, move the cursor to “F-CCCS”, and press **ENTER** to display the element parameter screen for a current-controlled current source. Enter element parameters until your screen appears as follows, making sure to add the node AMMETER when selecting the (+) node of FCE:

```

F-CCCS
→ELEMENT: FCE
NODE (+): AMMETER
NODE (-): EMITTER
CONTROL SOURCE: VBE
CURRENT GAIN: 100
MAIN ←STK PRINT VIEW FONT UP

```


This completes Step ②⑥. Press **UP** to return to the Element Manager.

Adding VAMTR and Node COLLECTOR

Press **ADD** to add a new element, move the cursor to “V–Source”, and press **ENTER** to display the element parameter screen for a voltage source. Enter element parameters until your screen appears as follows, making sure to add the node COLLECTOR when selecting the (+) node of VAMTR:

```
V-Source
→ELEMENT: VAMTR
NODE (+): COLLECTOR
NODE (-): AMMETER
DC VALUE (V): 0
AC VALUE (V): (0,0)
TRANSIENT: NONE
MAIN →STR PRINT VIEW FONT UP
```

This completes Step ②⑦. Press **UP** to return to the Element Manager.

Adding Resistor RC

Press **ADD** to add a new element, leave the cursor at “R–Resistor”, and press **ENTER** to display the element parameter screen for a resistor. Enter element parameters until your screen appears as follows:

```
R-Resistor
→ELEMENT: RC
NODE (+): BATTERY
NODE (-): COLLECTOR
VALUE (Ω): 10000
MAIN →STR PRINT VIEW FONT UP
```

This completes Step ②⑧. Press **UP** to return to the Element Manager.

Adding RL, CL and Node OUTPUT

Press **ADD** to add a new element, leave the cursor at “R–Resistor”, and press **ENTER** to display the element parameter screen for a resistor. Enter element parameters until your screen appears as follows, making sure to add the node OUTPUT when selecting the (+) node of RL:

```
R-Resistor
→ELEMENT: RL
NODE (+): OUTPUT
NODE (-): GROUND
VALUE (Ω): 1000
MAIN →STR PRINT VIEW FONT UP
```

Press **UP** to return to the Element Manager.

Press **ADD** to add a new element, move the cursor to “C–Capacitor”, and press **ENTER** to display the element parameter screen for a capacitor. Enter element parameters until your screen appears as follows:

```

      C-Capacitor
→ELEMENT: CL
NODE (+): COLLECTOR
NODE (-): OUTPUT
VALUE (F): .000001
INITIAL (V): 0
MAIN →STR PRINT VIEW FONT UP
```

This completes Step ②⑨. Press **UP** to return to the Element Manager.

Adding VBATT

Press **ADD** to add a new element, move the cursor to “V–Source”, and press **ENTER** to display the element parameter screen for a voltage source. Enter element parameters until your screen appears as follows:

```

      V-Source
→ELEMENT: VBATT
NODE (+): BATTERY
NODE (-): GROUND
DC VALUE (V): 20
AC VALUE (V): (0,0)
TRANSIENT: NONE
MAIN →STR PRINT VIEW FONT UP
```

This completes Step ②⑩. Press **UP** to return to the Element Manager.

This completes all of Step ②. Press **UP** to return to the Main menu.

Drawing the Schematic of BJTDC

Third, the circuit schematic can be drawn. This step can occur at any point after the circuit has been fully specified (steps ① and ②).

You will follow these steps:

- ① Set the node coordinates.
- ② Automatically generate the element coordinates and orientations.
- ③ View the partial schematic to check element locations.
- ④ Adjust element coordinates and orientations to optimize schematic.
- ⑤ Automatically generate the wires.

- ⑥ View the finished schematic.

At the Main menu, move the pointer to “Graphics Manager” and press **ENTER**.

Setting Node Coordinates

Make sure the pointer is at “Set Node Coordinates” and press **ENTER**. The circuit nodes will be displayed. Enter node coordinates until your screen appears as follows:

Nodes:		
(0,1) SOURC	(0,0) GND	(0,2) INPUT
(1,2) BASE	(5,3) BATT	(2,1) EMITT
(3,2) AMME	(4,2) COLLE	(5,2) OUTPUT
MAIN	EDIT	CLEAR UP

This completes Step ③①. Press **UP** to return to the Graphics Manager.

Setting Element Coordinates and Orientations

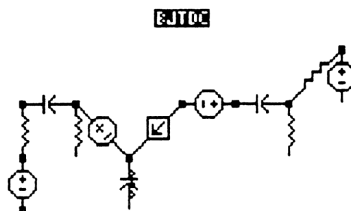
Move the pointer to “Set Elmt Coordinates” and press **ENTER**. The circuit elements will be displayed. Press **AUTO** to automatically generate the element coordinates and orientations from the node coordinates. The elements should be located as follows:

Elements:		
(0,1).1 VS	(0,1).5 RS	(0,2).7 CS
(5,3).2 RB1	(1,2).1 RB2	(1,2).8 VBE
(2,1).1 RE	(2,1).1 CE	(3,2).2 FCE
(4,2).3 VBM	(5,3).2 RC	(5,2).1 RL
(4,2).7 CL	(6,3).1 VEM	
MAIN	EDIT	AUTO CLEAR UP

This completes Step ③②. Press **UP** to return to the Graphics Manager.

Viewing the Partial Schematic

Move the pointer to “View Schematic” and press **ENTER** (Because the schematic is larger than the screen, it will be automatically centered on the screen, and the cursor keys will be activated for scrolling.):



Note that two of the elements overlap (CE needs to be moved away from RE.), some of the elements point diagonally between their nodes (RC erroneously appears to touch node OUTPUT.), and one of the elements needs to be moved (RB1 is hidden beneath RC.). When you have finished viewing the schematic, press **ENTER** to return to the Graphics Manager.

This completes Step 33.

Adjusting Element Coordinates and Orientations

Move the pointer to “Set Elmt Coordinates” and press **ENTER**. The circuit elements will be displayed. Adjust the coordinates and orientations of elements RB1, CE, and RC, until your screen appears as follows:

Elements:					
(0.1) 1 VS	(0.1) 5 RE	(0.2) 7 CS			
(1.3) 1 RB1	(1.2) 1 RB2	(1.2) 8 VBE			
(2.1) 1 RE	(3.1) 1 CE	(3.2) 2 FCE			
(4.2) 3 VWM	(4.3) 1 RC	(5.2) 1 RL			
(4.2) 7 CL	(6.3) 1 VEM				
MAIN	EDIT	AUTO	CLEAR	UP	

This completes Step 34. Press **UP** to return to the Graphics Manager.

Setting Wire Coordinates

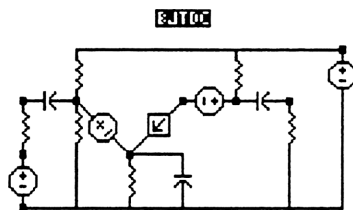
Move the pointer to “Set Wire Coordinates” and press **ENTER**. No wires currently exist. Press **AUTO** to automatically generate the wires necessary to fix “loose” connections in the circuit. The wires should be located as follows:

Wires:					
(0.0) (6.0)	(0.0) (1.0)	(0.0) (5.0)			
(0.0) (3.0)	(0.0) (2.0)	(1.3) (6.3)			
(1.1) (1.0)	(2.1) (3.1)	(4.3) (6.3)			
(5.1) (5.0)	(6.2) (6.0)				
ADD	DEL	EDIT	AUTO	CLEAR	UP

This completes Step 35. Press **UP** to return to the Graphics Manager.

Viewing the Finished Schematic

Move the pointer to “View Schematic” and press **ENTER**. You will be prompted to update the schematic. Press **YES** to utilize the modified coordinates and the wires to rebuild the schematic, which will then appear (Because the schematic is larger than the screen, it will be automatically centered on the screen, and the cursor keys will be activated for scrolling.):



When you have finished viewing the schematic, press **ENTER** to return to the Graphics Manager.

This completes Step ③⑥ and therefore all of Step ③. Press **UP** to return to the Main menu.

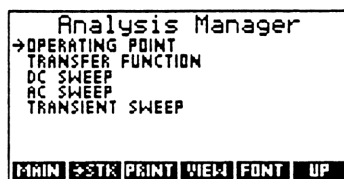
Analyzing BJTDC

Fourth, the circuit can be analyzed and the resulting data viewed, plotted, and saved. This step can occur at any point after the circuit has been fully specified (steps ① and ②).

You will perform the following analyses:

- ① Operating Point (OP) analysis to find the DC steady-state voltages and currents in order to investigate the cut-off and saturation conditions.
- ② DC Sweep (DC) analysis to investigate the stability of the VBE bias point and to determine the effects of variation there on the collector current (a sensitivity parameter: $\partial I / \partial V_{BE}$). Use the ammeter VAMTR to find the collector current. Sweep VBE from 0.60 V to 0.75 V every 0.025 V.

At the Main menu, move the pointer to “Analysis Manager” and press **ENTER**:

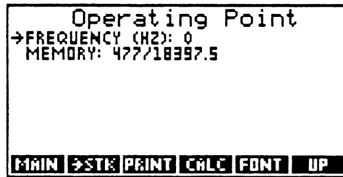


Operating Point (OP) Analysis

Analysis: Determine the DC steady-state operation point of circuit BJTDC in order to investigate the cut-off

Performing an OP Analysis

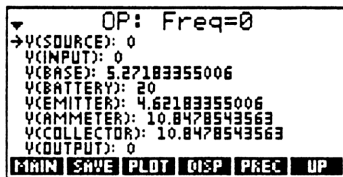
Make sure the pointer is at “Operating Point” and press **ENTER**. There will be a delay (~ 60 sec) while the circuit is prepared, and then the analysis parameter screen appears:



Press **CALC**. There will be a delay (~ 15 sec) while the analysis is performed.

Viewing the Results (Data View)

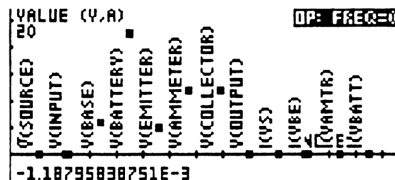
When the calculation completes, the OP results will be displayed in the Output Manager Data View, which (in this case) displays voltage and current data:



From the OP analysis data, look for cut-off and saturation conditions by determining the collector-emitter voltage drop $V(\text{COLLECTOR}) - V(\text{EMITTER}) = 10.82 - 4.62 = 6.23 > 0$. Since this is positive and $I(VBE) > 0$, the NPN BJT is in the normal operating region.

Plotting Node Values (Node Plot)

We can determine the same information graphically. Press **PLOT** to plot the OP results, and at the prompt, press **ENTER** to accept the default vertical range. The following plot appears (Because the plot is wider than the screen, it will be automatically centered on the screen, and the cursor keys will be activated for scrolling.):



This node plot displays the voltage or current of each output node, with a corresponding label to the left of each point. Visually, you can clearly see that

$V(\text{COLLECTOR}) > V(\text{EMITTER})$, although because of the scale, it is difficult to determine if $I(VBE) > 0$.

When you have finished viewing the plot, press **ATTN** to return to the Output Manager Data View.

Saving the Results

Press **NXT** **MEM** to determine the free memory required to save the OP analysis data. If you have enough free memory, press **NXT** **SAVE**, and at the prompt press **ENTER** to accept the default name. The OP analysis data will be stored into the file 'BJTDC.OP' in the 'fileS' subdirectory of the 'SPICE48D' directory.

Further OP Analysis

If you *decrease* the value of resistor RB2 from 20K Ω down to 1.5K Ω and perform an OP analysis, you will see a cut-off case where $I(VBE) < 0$. In an actual transistor, you would see $I(VBE) = 0$.

If you *increase* the value of resistor RB2 from 20K Ω up to 54K Ω and perform an OP analysis, you will see a saturation case where $V(\text{COLLECTOR}) - V(\text{EMITTER}) < 0$. In an actual transistor, this value would be 0.

This completes Step 4①. Press **UP** to return to the Analysis Manager.

DC Sweep (DC) Analysis

Analysis: Investigate the VBE bias point by sweeping VBE from 0.60 V to 0.75 V every 0.025 V. Plot the collector current $I(\text{VAMTR})$ as a function of VBE, to examine the stability of the transistor with respect to the dramatic voltage changes at VBE. Determine the slope of that plot at the value 0.65 V for VBE (as in the previous OP analysis), which will determine *analytically* the variation in collector current as a function of the voltage VBE.

Performing a DC Analysis

Make sure the pointer is at "DC Sweep" and press **ENTER**. The analysis parameter screen appears:

DC Sweep	
→SOURCE:	PRESS ENTER
START (V,A):	PRESS ENTER
STOP (V,A):	PRESS ENTER
STEP (V,A):	PRESS ENTER
MEMORY:	???/16974.5
MAIN →STEP PRINT CALC FONT UP	

Set the source to VBE, set the start value to 0.60 V, set the stop value to 0.75 V, set the step value to 0.025 V, and press **CALC**. There will be a delay (~ 120 sec) while the analysis is performed.

Viewing the Results (Node View)

When the calculation completes, the DC results will be displayed in the Output Manager Node View, which displays all output nodes (For more information, refer to “Node View” in Chapter 9.):

DC: Nodes		
V(SOURCE)	V(INPUT)	V(BASE)
V(BATTERY)	V(EMITTER)	V(AMMETER)
V(COLLECTOR)	V(OUTPUT)	I(VS)
I(VBE)	I(VAMTR)	I(VBATT)
MAIN MARK PLOT NODE SRC UP		

The output nodes for the DC analysis include V(SOURCE), V(INPUT), V(BASE), V(BATTERY), V(EMITTER), V(AMMETER), V(COLLECTOR), V(OUTPUT), I(VS), I(VBE), I(VAMTR), and I(VBATT), for a range (sweep) of values of source VBE from 0.60 V to 0.75 V every 0.025 V.

Viewing the Results (Source Value View)

These DC analysis results include the same values as were just calculated in the previous OP analysis. Press **SRC** to switch to the Output Manager Source Value View, which displays all swept values of the source VCE (For more information, refer to “Source Value View” in Chapter 9.):

DC: VBE Values		
.6	.625	.65
.675	.7	.725
.75		
MAIN MARK PLOT NODE SRC UP		

Viewing the Results (Data View)

Examine the node voltages and currents at value .65 for VBE. Press **▶▶** to move the cursor to .65 and press **ENTER** to switch to the Output Manager Data View, which (in this case) displays voltage and current data (For more information, refer to “Data View” in Chapter 9.):

DC: VBE=.65	
▼	
→V(SOURCE):	0
V(INPUT):	0
V(BASE):	5.27183355006
V(BATTERY):	20
V(EMITTER):	4.62183355006
V(AMMETER):	10.8478543563
V(COLLECTOR):	10.8478543563
V(OUTPUT):	0
MAIN SAVE PLOT DISP PREC UP	

This is identical to the earlier OP analysis data.

When you have finished viewing the data, press **UP** to return to the Output Manager Source Value View and press **NODE** to switch to the Output Manager Node View.

Plotting I(VAMTR) vs. V(VBE) (Value Plot, Equation)

Move the cursor to the output node I(VAMTR) and press **PLOT**, then **ENTER** to select an equation plot. (An equation plot will allow us to determine the slope at any point, whereas a point or line plot will not.) Then press **ENTER** to accept the default horizontal range.

The vertical range will default to a minimum value of 0, but because the collector current will vary only slightly, you should adjust the vertical range to vary from 0.75 mA to 1 mA. To do this, press **ATN** 0.00075 **SPC** 0.001:

```

PRG
{ HOME SPARCOM SPICE4BD BJTDC }
Edit vertical range:
→ min max
( reals, min<max )
.00075 .001
<SKIP SKIP> <DEL DEL> INS <STX>

```

Press **ENTER** to accept the vertical range. Next, you will be prompted to edit the plot resolution:

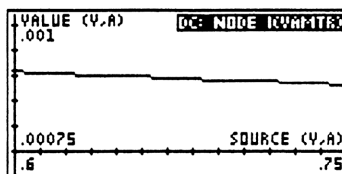
```

PRG
{ HOME SPARCOM SPICE4BD BJTDC }
Edit plot resolution:
→ #binary for pixels
→ real for user-units
#1
<SKIP SKIP> <DEL DEL> INS <STX>

```

Because the equation plot actually creates piecewise linear representations of the data, algebraic functions can be plotted. If you expect the data to be linear, a larger plot resolution can be tolerated without adversely affecting the display, whereas if the data is non-linear, a smaller plot resolution should be used to guarantee an accurate graph. In this example, the data should be quite linear, so press 0 **ENTER** to increase the plot resolution to 10 pixels. (For more information, refer to “Value Plots” in Chapter 9.)


The following plot appears:

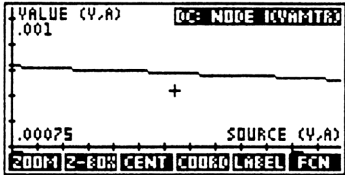



This value plot displays the output node I(VAMTR), the collector current, as a function of source VBE voltage.

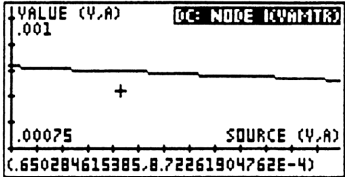
As you see, the circuit appears quite stable.


Determining the Slope

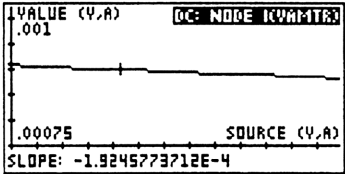
Because this is an equation plot, you can utilize the built-in Graphics Environment of your HP 48SX to further analyze the results. Press  to activate the Graphics Environment:





Press **COORD** to turn on the coordinate display, and move the cursor to the left to find the closest point to 0.65 V (press  21 times):



Press  to turn off the coordinate display, and press **FCN** to access the function analysis menu. Press **SLOPE** to calculate the slope at this point. After a brief delay (~ 15 sec), the slope will be displayed:



The slope value is $-0.19 \text{ mA} / \text{V}$. Because the plot is so linear, this slope value can be taken to indicate in real terms that for each 10 mV drop in the base-emitter voltage, the collector current will drop by $\sim 2 \text{ }\mu\text{A}$. This is the sensitivity parameter $\partial I / \partial V_{BE}$. (The slope value will automatically be copied to the stack of your HP 48SX.)

Press  to clear the slope value display. When you have finished viewing and analyzing the plot, press  to return to the Output Manager Node View. (For more information, refer to “Value Plot” in Chapter 9 of this manual and refer to

Chapter 18 of the HP 48SX Owner's Manual, "Basic Plotting and Function Analysis.")

Saving the Results

Press **NXT** **MEM** to determine the free memory required to save the DC analysis data. If you have enough free memory, press **SAVE**, and at the prompt press **ENTER** to accept the default name. The DC analysis data will be stored into the file 'BJTDC.DC' in the 'fileS' subdirectory of the 'SPICE48D' directory. (For more information, refer to Chapter 8, "Data File Manager.")

This completes Step 4②. Press **UP** to return to the Analysis Manager.

This completes all of Step 4. Press **UP** to return to the Main menu.

This completes all of Example: Bipolar Junction Transistor, DC Model.

Chapter 3

Circuit Manager



The Circuit Manager is used to add new circuits, delete unwanted circuits, pack circuits to free memory, or transfer circuits to and from a PC, Macintosh, or other computer. It is also where you select the current circuit to work on with Spice48.

This chapter covers:

- ☐ Using the Circuit Manager
- ☐ Selecting the Current Circuit
- ☐ Adding a Circuit
- ☐ Deleting a Circuit
- ☐ Packing a Circuit
- ☐ Transferring a Circuit

Using the Circuit Manager

To get to the Circuit Manager, follow these steps:

- ❶ Press  **LIBRARY** to display all libraries available to your HP 48SX.
- ❷ Find and press **SPICE** to enter the Spice48 Pac library directory.
- ❸ Press the first softkey, **SPICE**, to start the Spice48 Pac.
- ❹ At the Main menu, make sure the pointer is at “Circuit Manager” and press .

Items in the Circuit Manager

The items listed in the Circuit Manager are the circuits currently stored in the 'SPICE48D' directory of your HP 48SX. Circuits that were archived to a computer and then deleted will not be shown. The following screen includes the two tutorial circuits (For more information, refer to Chapter 2, “Spice48 Tutorial.”):



After further use of the Spice48 Pac, the Circuit Manager will also include your circuits, or if you have no circuits stored, the screen will display, “Press ADD.”

At the top of the screen is indicated the free memory in your HP 48SX, and next to each circuit is indicated the amount of free memory it currently occupies in the 'SPICE48D' directory. Some circuits may have the message “(Packed)” displayed next to them, which indicates that they are newly created or recently packed, and contain no temporary calculation or graphics files.

One of the circuits will have the message “(Current)” displayed next to it: this is the currently loaded circuit in the Spice48 Pac.

Summary of Operations

Screen	Softkeys
Circuit Manager	ADD DEL PACK XFER FONT UP

Key	Action
ADD	Prompts for the name of a new circuit and initializes it.
DEL	Deletes the circuit currently selected by the pointer.
FONT	Toggles between the small and large fonts.
PACK	Packs the circuit currently selected by the pointer.
XFER	Transfers a circuit to or from a computer.
UP	Moves up one level in the menu structure, returning to the Main menu.
	Quits the Spice48 Pac to the HP 48SX stack.
	Selects the circuit currently selected by the pointer as the current circuit for Spice48 Pac operations.
	Displays the entire text of an item too wide to fit on the screen, up to one entire screen size. If the item fits on the screen, this key is non-functional.
	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Selecting the Current Circuit

Before you can modify a circuit or perform analyses on it, you must make sure it is the current circuit. To set the current circuit, go to the Circuit Manager, move the pointer to the desired circuit, and press **ENTER**. This loads in that circuit’s data and labels the circuit as “(Current)” in the Circuit Manager. After selecting a circuit, you will be returned to the Main menu.

Adding a new circuit or transferring a circuit to your HP 48SX from a computer will automatically set that circuit to be the current circuit. Deleting the current circuit automatically resets the current circuit to the first circuit in the Circuit Manager.

Adding a Circuit

To add a new circuit, go to the Circuit Manager and press **ADD**. You will be prompted for the name of the circuit to add:



The screenshot shows a terminal window titled "PRG" with a header line "{ HOME SPARCOM SPICE48D }". Below the header, the text "Enter circuit name:" is displayed. A cursor points to the word "name". At the bottom of the screen, a row of function keys is visible: **←SKIP SKIP→**, **←DEL DEL→**, **INS**, and **↑STK**.

Type in a name for the new circuit and press **ENTER**, or to abort the add operation, press **ATTN** to clear the command line (if necessary), and then press **ENTER** or **ATTN** to return to the Circuit Manager without entering a new circuit. The circuit name must be a valid HP 48SX variable name, because it will be used to create a subdirectory of the 'SPICE48D' directory to store the circuit data. For this reason, no spaces are allowed, and the name must begin with a letter. (For more information on naming conventions, refer to Chapter 6 of the HP 48SX Owner's Manual, "Variables and the VAR Menu." For more information on how to use the editing softkeys, refer to "Text Editing" in Chapter 1 of this manual.)

After pressing **ENTER**, a new circuit directory will be created, and you will be returned to the Circuit Manager. The pointer will be set to the newly created circuit, which will be labeled as both "(Packed)" and "(Current)." Newly created circuits require 151 bytes of free memory.

Deleting a Circuit

To delete a circuit, go to the Circuit Manager, move the pointer to the desired circuit, and press **DEL**. You will be prompted to verify the delete operation by pressing **YES** to confirm or **NO** or **ATTN** to abort. If you confirm the delete operation, that circuit directory will be deleted and all the data for that circuit erased. Deleting the current circuit automatically resets the current circuit to the first circuit in the Circuit Manager.

Packing a Circuit

To pack a circuit, go to the Circuit Manager, move the pointer to the desired circuit, and press **PACK**.

Packing a circuit deletes a number of temporary global variables that reside in a circuit's directory and assist with calculations and graphics operations. Packing does not affect any of the essential global variables that describe the circuit, including graphics coordinates.

The disadvantage of packing a circuit is that the next time an analysis is performed, the circuit will have to be “prepared,” and all the temporary global variables recreated for calculations—this can be time-consuming. For that reason, we recommend that you leave unpacked the circuits you use regularly. Note that a packed circuit will automatically be unpacked by Spice48 when necessary.

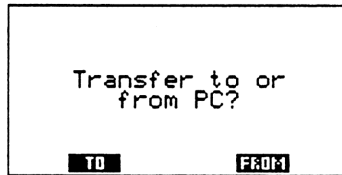
The advantage of packing a circuit is that it frees memory for new circuits or large calculations. It also shrinks a circuit before transferring it to a computer. The savings may be substantial: we have encountered cases where a fully unpacked circuit with a graphics schematic takes up over 15K but the packed form is less than 2K.

Transferring a Circuit

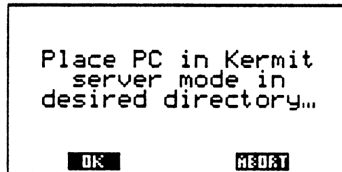
A circuit can be transferred either to or from a PC, Macintosh, or other computer with Kermit software and an HP 48SX cable.

Transferring a Circuit to a Computer

To download a circuit to a computer, go to the Circuit Manager, move the pointer to the desired circuit, and press **XFER**:



At this prompt, press **TO**:



Now, you must prepare your computer to receive a file with Kermit protocol. For a PC, this involves running MS-KERMIT. For a Macintosh, this involves running a terminal emulator supporting Kermit or running the public domain version of Kermit. For other computers, this involves running some public domain version of Kermit. In any case, you must set the transfer directory on the computer to the directory to which you want the circuit transferred. Then, you must either prepare the computer to receive a single file or place the computer in Kermit server mode. When you have done this, press **OK** to download the circuit. To abort the transfer, press **ABORT** or **ATTN**. After the transfer has completed, you may cancel Kermit server mode on your computer and quit the Kermit program.

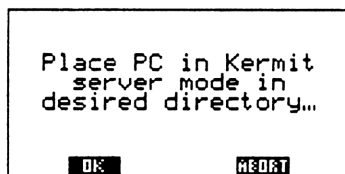
If you successfully download a circuit to your computer from your HP 48SX, it will be stored into an ASCII text file in the transfer directory. The circuit directory will not be affected on your HP 48SX, and if you have transferred the circuit for archival purposes, you can now safely delete it by pressing **DEL**. (For more information on deleting a circuit, refer above to “Deleting a Circuit.”)

Transferring a Circuit from a Computer

To upload a circuit from a computer, go to the Circuit Manager, press **XFER**, and press **FROM**. You will be prompted to enter the name of the circuit you wish to download from the computer:



Press **ENTER** when you have finished typing the name:



Now, you must prepare your computer to be a Kermit server. For a PC, this involves running MS-KERMIT. For a Macintosh, this involves running a terminal emulator supporting Kermit or running the public domain version of Kermit. For other computers, this involves running some public domain version of Kermit. In any case, you must set the transfer directory on the computer to the directory in which the circuit you want resides. Then, you must place the computer in Kermit server mode. When you have done this, press **OK** to upload the circuit. To abort the transfer, press **ABORT** or **ATTN**. After the transfer has completed, you may cancel Kermit server mode on your computer and quit the Kermit program.

If you successfully upload a circuit to your HP 48SX from your computer, it will be stored into a subdirectory of the 'SPICE48D' directory, and will automatically become the current circuit:



Chapter 4

Element Manager



The Element Manager is used to add, delete, and edit elements in the current circuit. (For more information on how to select the current circuit, refer to “Selecting the Current Circuit” in Chapter 3.)

This chapter covers:

- ☐ Using the Element Manager
- ☐ Adding an Element
- ☐ Deleting an Element
- ☐ Editing an Element
- ☐ Element Parameter Entry Tips
- ☐ R–Resistor
- ☐ C–Capacitor
- ☐ L–Inductor
- ☐ K–Coupled (Coupled Inductors)
- ☐ V–Source (Independent Voltage Source)
- ☐ I–Source (Independent Current Source)
- ☐ E–VCVS (Voltage-Controlled Voltage Source)
- ☐ F–CCCS (Current-Controlled Current Source)
- ☐ G–VCCS (Voltage-Controlled Current Source)
- ☐ H–CCVS (Current-Controlled Voltage Source)
- ☐ Transient Sources
- ☐ Pulse Transient
- ☐ PWL Transient
- ☐ Function Transient

Using the Element Manager

To get to the Element Manager, follow these steps:

- ❶ Press   to display all libraries available to your HP 48SX.
- ❷ Find and press **SPICE** to enter the Spice48 Pac library directory.
- ❸ Press the first softkey, **SPICE**, to start the Spice48 Pac.

- ❶ At the Main menu, make sure the pointer is at “Element Manager” and press **ENTER**.

Items in the Element Manager

The items listed in the Element Manager are the elements in the current circuit. The following list of elements is from tutorial circuit BJTDC (For more information, refer to Chapter 2, “Spice48 Tutorial.”):

Elements:		
VS	RS	CS
BB1	BB2	VBE
RE	CE	FCE
VBMTR	RC	RL
CL	VBATT	
ADD	DEL	EDIT
		UP

If the current circuit has no elements, the screen will display, “Press ADD.”

Summary of Operations

Screen	Softkeys
Element Manager	ADD DEL EDIT UP

Key	Action
ADD	Adds a new element to the current circuit.
DEL	Deletes the element currently selected by the cursor.
EDIT	Edits the element currently selected by the cursor.
UP	Moves up one level in the menu structure, returning to the Main menu.
ATTN	Quits the Spice48 Pac to the HP 48SX stack.
ENTER	Edits the element currently selected by the cursor, or if no elements exist, adds a new element to the current circuit
ON PREV	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Adding an Element

To add a new element to the current circuit, go to the Element Manager and press **ADD**. You will be prompted for the type of element to add:

Pick element type:		
R-RESISTOR	C-CAPACITOR	L-INDUCTOR
K-COUPLED	V-SOURCE	I-SOURCE
E-VCVS	F-CCCS	G-VCCS
H-CCVS		
MAIN		UP

Move the cursor to the desired element type and press **ENTER**. (Instead of pressing **ENTER**, you can press **UP** to abort the operation, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.)

After you have selected the element type to add to the current circuit, you will be presented with a screen containing a number of parameters required to specify the chosen type of element (the “element parameter screen”). The exact parameters for each of the various types of elements are explained in detail in the sections below.

Deleting an Element

To delete an element, go to the Element Manager, move the cursor to the desired element, and press **DEL**. You will be prompted to verify the delete operation by pressing **YES** to confirm or **NO** or **ATTN** to abort. If you confirm the delete operation, the current circuit will be searched for references to the element, and if no references are found, the element will be deleted. If the element is referenced by any elements in the current circuit, you will first have to either delete those elements or else change the element references to some other element.

Editing an Element

To edit an element, go to the Element Manager, move the cursor to the desired element, and press **ENTER** or **EDIT**. You will be presented with the element parameter screen for the current element, which will display the current values of the parameters for this element. To edit the value of a parameter, move the pointer to it and press **ENTER**. After editing the value, press **ENTER** to accept the new value and return to the element parameter screen, or to abort changing the parameter value, press **ATTN** to clear the command line (if necessary) and then

press **ENTER** or **ATTN** to return to the element parameter screen without changing the parameter value. When you have finished editing the current element, press **UP** to return to the Element Manager—the updated element will be stored automatically.

For specific information about each type of element, refer to the appropriate section below.

Element Parameter Entry Tips

Here are some useful things to know about entering or editing element parameters:

- ☐ To enter or edit the value of a parameter, move the pointer to it and press **ENTER**. After entering or editing the value of a parameter, press **ENTER** to accept the new value and return to the element parameter screen.
- ☐ Once you have begun entering or editing a parameter value, to abort the change, press **ATTN** to clear the command line (if necessary) and then press **ENTER** or **ATTN** to return to the analysis parameter screen without changing the parameter value.
- ☐ If you press **UP**, **MAIN**, or **ATTN** while the new element is only partially specified, you will be asked to verify that you want to abandon the partial entry. Press **YES** to abandon the entry, or **NO** or **ATTN** to continue specifying the new element.
- ☐ You only need to enter values for those parameters that initially display, “Press ENTER.” Other parameters (e.g., the initial value of a capacitor, the transient type of a voltage source) will be given default values (e.g., 0 and NONE, respectively) which you may modify if you wish.
- ☐ When you are entering a numeric value, make use of the softkeys to append the desired exponent to the value (e.g., type 3 and then press **MICRO** to enter the value .000003). More exponent softkeys can be accessed by pressing **[NXT]**.

Selecting a Node during Parameter Entry

To specify each type of circuit element (except for the coupled inductor type), you must select both positive (+) and negative (–) nodes. When you press **ENTER** to select the (+) or (–) node for the current element, you will be presented with a list of all of the nodes in the current circuit. To pick one of the nodes, move the

cursor to the desired node and press **ENTER**. (Instead of pressing **ENTER**, you can press **UP** to abort the operation, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.) If you are *changing* the (+) or (–) node of the current element, the cursor will automatically be placed on the current node; otherwise, the cursor will be placed on the first node in the list.

If the node you wish to use as the (+) or (–) node of the current element has not yet been added to the current circuit, press **ADD**, to enter the node(s) you wish to add, press **ENTER** to accept the node(s) entry, move the cursor to the desired new node, and press **ENTER** to select that node and return to the element parameter screen. (For more information on adding nodes, refer to Chapter 6, “Node Manager.”)

Selecting an Element during Parameter Entry

To specify certain types of circuit elements (coupled inductor elements and both types of current-controlled sources), you must select other circuit elements.

When you press **ENTER** to select one of the inductors in a coupled inductor element, or to select the controlling source for a current-controlled source, you will be presented with a list of all of the elements in the current circuit. To pick one of the elements, move the cursor to the desired element and press **ENTER**. (Instead of pressing **ENTER**, you can press **UP** to abort the operation, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.) You will only be allowed to select an appropriate type of element (e.g., inductors or sources, as appropriate). If you are *changing* the inductor or controlling source of the current element, the cursor will automatically be placed on the current inductor or controlling source; otherwise, the cursor will be placed on the first element in the list.

If the element you wish to use as an inductor for a coupled inductor element or as the controlling source for a current-controlled source has not yet been added to the current circuit, press **ADD**, to add a new element, select the type of element you wish to add (inductor or source), specify the new element at its element parameter screen, press **UP** to accept the new element (and return to the list of circuit elements), move the cursor to the desired new element, and press **ENTER** to select that element and return to the element parameter screen for the current element. (For more information on adding elements, refer below to the appropriate section.)

Summary of Operations

Screen	Softkeys					
Element/ Transient Parameter	MAIN	STK	PRINT	VIEW	FONT	UP

Key	Action
FONT	Toggles between the small and large fonts.
MAIN	Returns to the Main menu. The element is saved automatically.
PRINT	Prompts for ONE or ALL to select items, and then sends those items to an IR printer.
-STK	Prompts for ONE or ALL to select items, and then copies those items to the stack. The items are placed in a list if ALL was chosen.
UP	Moves up one level in the menu structure. At an element parameter screen, completes element entry and returns to the Element Manager. At a transient parameter screen, completes transient entry and returns to the element parameter screen. The element or transient is saved automatically.
VIEW	Displays the entire text of an item too wide to fit on the screen, up to one entire screen size. If the item fits on the screen, this key is non-functional.
ATTN	Quits the Spice48 Pac to the HP 48SX stack. The element is saved automatically.
ENTER	Prompts for the value of the parameter currently selected by the pointer.
ON - PREV	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

R-Resistor

Upon selecting an element of this type, the following element parameter screen appears:

```

R-Resistor
→ELEMENT: PRESS ENTER
NODE (+): PRESS ENTER
NODE (-): PRESS ENTER
VALUE (Ω): PRESS ENTER

MAIN →STK PRINT VIEW FONT UP

```


You should now enter or edit the parameter values. When you have finished, press **UP** to return to the Element Manager, and the element will be saved automatically.

Here are specific descriptions of these parameters:

- ☐ **Element:** Name must begin with “R” and include no spaces. Possible names include: “R1,” “RTEST,” “RloadBtoC,” etc.
- ☐ **Node (+):** Selected from the Node Manager. (For more information, refer above to “Selecting a Node during Parameter Entry.”)
- ☐ **Node (–):** Selected from the Node Manager. The (+) and (–) nodes must be different.
- ☐ **Value (Ω):** Must be a real number, will automatically have units of ohms.

C-Capacitor

Upon selecting an element of this type, the following element parameter screen appears:



```
C-Capacitor
→ELEMENT: PRESS ENTER
NODE (+): PRESS ENTER
NODE (-): PRESS ENTER
VALUE (F): PRESS ENTER
INITIAL (V): 0
MAIN | ← | STK | PRINT | VIEW | FONT | UP
```

You should now enter or edit the parameter values. When you have finished, press **UP** to return to the Element Manager, and the element will be saved automatically.

Here are specific descriptions of these parameters:

- ☐ **Element:** Name must begin with “C” and include no spaces. Possible names include: “C1,” “CIN,” “Cload,” etc.
- ☐ **Node (+):** Selected from the Node Manager. (For more information, refer above to “Selecting a Node during Parameter Entry.”)
- ☐ **Node (–):** Selected from the Node Manager. The (+) and (–) nodes must be different.

- ☐ **Value (F):** Must be a real number, will automatically have units of Farads.
- ☐ **Initial (V):** (Initial capacitor voltage). Must be a real number, will automatically have units of volts, defaults to 0. Non-zero values are used when performing a Transient Sweep (TR) analysis with the “Use OP or IC” parameter set to IC to use element initial conditions.

L-Inductor

Upon selecting an element of this type, the following element parameter screen appears:

L-Inductor

→ELEMENT: PRESS ENTER

NODE (+): PRESS ENTER

NODE (-): PRESS ENTER

VALUE (H): PRESS ENTER

INITIAL (A): 0

MAIN ← STK PRINT VIEW FONT UP

You should now enter or edit the parameter values. When you have finished, press **UP** to return to the Element Manager, and the element will be saved automatically.

Here are specific descriptions of these parameters:

- ☐ **Element:** Name must begin with “L” and include no spaces. Possible names include: “L3,” “LOUT,” “Lparallel,” etc.
- ☐ **Node (+):** Selected from the Node Manager. (For more information, refer above to “Selecting a Node during Parameter Entry.”)
- ☐ **Node (-):** Selected from the Node Manager. The (+) and (-) nodes must be different.
- ☐ **Value (H):** Must be a real number, will automatically have units of Henrys.
- ☐ **Initial (A):** (Initial inductor current). Must be a real number, will automatically have units of amperes, defaults to 0. Non-zero values are used when performing a Transient Sweep (TR) analysis with the “Use OP or IC” parameter set to IC to use element initial conditions.

K-Coupled (Coupled Inductors)

This type of circuit “element” is unusual in that it does not physically exist as an element but instead links two existing inductors. The Spice48 model for coupled inductors is in the form of inductors and a coupling factor (k) which describes the flux linkage between the inductors. This model can be related to measured values of self (L_1, L_2) and mutual (M) inductances by:

$$M = k\sqrt{L_1L_2}$$

where $0 \leq k \leq 1$. $k = 0$ corresponds to uncoupled coils; $k = 1$ corresponds to complete coupling. The positive terminals of coupled inductors in Spice48 are the “dotted” terminals of the “dot convention” of circuits. Thus, $k < 0$ is equivalent to reversing the dots on one coil and making the sign of the mutual inductance negative. Finally, $k > 1$ corresponds to a “flux multiplier,” not physically realizable, but of possible interest in rare cases.

Upon selecting an element of this type, the following element parameter screen appears:



```

K-Coupled
→ELEMENT: PRESS ENTER
INDUCTOR 1: PRESS ENTER
INDUCTOR 2: PRESS ENTER
COUPLING FACTOR: PRESS ENTER

MAIN →STK PRINT VIEW FONT UP

```

You should now enter or edit the parameter values. When you have finished, press **UP** to return to the Element Manager, and the element will be saved automatically.

Here are specific descriptions of these parameters:

- ☐ **Element:** Name must begin with “K” and include no spaces. Possible names include: “K1,” “K1to2,” “KL1L2,” etc.
- ☐ **Inductor 1:** Selected from the Element Manager. (For more information, refer above to “Selecting an Element during Parameter Entry.”)
- ☐ **Inductor 2:** Selected from the Element Manager. Inductors 1 and 2 must be different.
- ☐ **Coupling Factor:** Must be a real number.

V-Source (Independent Voltage Source)

Upon selecting an element of this type, the following element parameter screen appears:



```
V-Source
→ELEMENT: PRESS ENTER
NODE (+): PRESS ENTER
NODE (-): PRESS ENTER
DC VALUE (V): 0
AC VALUE (V): (0,0)
TRANSIENT: NONE

MAIN STK PRINT VIEW FONT UP
```

You should now enter or edit the parameter values. When you have finished, press **UP** to return to the Element Manager, and the element will be saved automatically.

Here are specific descriptions of these parameters:

- ☐ **Element:** Name must begin with “V” and include no spaces. Possible names include: “V1,” “V12volts,” “VSRC,” etc.
- ☐ **Node (+):** Selected from the Node Manager. (For more information, refer above to “Selecting a Node during Parameter Entry.”)
- ☐ **Node (-):** Selected from the Node Manager. The (+) and (-) nodes must be different.
- ☐ **DC Value (V):** Must be a real number, will automatically have units of volts, defaults to 0.
- ☐ **AC Value (V):** Must be a complex number, will automatically have units of volts, defaults to (0,0), can be entered as either (real,imaginary) or (magnitude,∠phase).
- ☐ **Transient:** Selected from NONE, PULSE, PWL, and FUNCTION, defaults to NONE. (For more information, refer below to “Transient Sources.”)

I-Source (Independent Current Source)

Upon selecting an element of this type, the following element parameter screen appears:



```

I-Source
→ELEMENT: PRESS ENTER
NODE (+): PRESS ENTER
NODE (-): PRESS ENTER
DC VALUE (A): 0
AC VALUE (A): (0,0)
TRANSIENT: NONE

MAIN >STK PRINT VIEW FONT UP

```

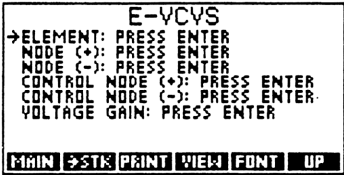
You should now enter or edit the parameter values. When you have finished, press **UP** to return to the Element Manager, and the element will be saved automatically.

Here are specific descriptions of these parameters:

- ☐ **Element:** Name must begin with “I” and include no spaces. Possible names include: “I2,” “I3amps,” “ISRC,” etc.
- ☐ **Node (+):** Selected from the Node Manager. (For more information, refer above to “Selecting a Node during Parameter Entry.”)
- ☐ **Node (-):** Selected from the Node Manager. The (+) and (-) nodes must be different.
- ☐ **DC Value (A):** Must be a real number, will automatically have units of amperes, defaults to 0.
- ☐ **AC Value (A):** Must be a complex number, will automatically have units of amperes, defaults to (0,0); can be entered as either (real,imaginary) or (magnitude,angle).
- ☐ **Transient:** Selected from NONE, PULSE, PWL, and FUNCTION, defaults to NONE. (For more information, refer below to “Transient Sources.”)

E-VCVS (Voltage-Controlled Voltage Source)

Upon selecting an element of this type, the following element parameter screen appears:



You should now enter or edit the parameter values. When you have finished, press **UP** to return to the Element Manager, and the element will be saved automatically.


Here are specific descriptions of these parameters:

- ☐ **Element:** Name must begin with “E” and include no spaces. Possible names include: “E0,” “E74,” “Enode7node4,” etc.
- ☐ **Node (+):** Selected from the Node Manager. (For more information, refer above to “Selecting a Node during Parameter Entry.”)
- ☐ **Node (-):** Selected from the Node Manager. The (+) and (-) nodes must be different.
- ☐ **Control Node (+):** Selected from the Node Manager.
- ☐ **Control Node (-):** Selected from the Node Manager. The control (+) and control (-) nodes must be different.
- ☐ **Voltage Gain:** Must be a real number. The gain relates the source voltage to the controlling voltage by:

$$V(+)-V(-)=Gain[V(C(+))-V(C(-))].$$

F-CCCS (Current-Controlled Current Source)

Upon selecting an element of this type, the following element parameter screen appears:



```
F-CCCS
→ELEMENT: PRESS ENTER
NODE (+): PRESS ENTER
NODE (-): PRESS ENTER
CONTROL SOURCE: PRESS ENTER
CURRENT GAIN: PRESS ENTER

MAIN ← → PRINT VIEW FONT UP
```

You should now enter or edit the parameter values. When you have finished, press **UP** to return to the Element Manager, and the element will be saved automatically.

Here are specific descriptions of these parameters:

- ☐ **Element:** Name must begin with “F” and include no spaces. Possible names include: “F0,” “F74,” “Fnode7node4,” etc.
- ☐ **Node (+):** Selected from the Node Manager. (For more information, refer above to “Selecting a Node during Parameter Entry.”)
- ☐ **Node (–):** Selected from the Node Manager. The (+) and (–) nodes must be different.
- ☐ **Control Source:** Must be an independent voltage source. Selected from the Element Manager. (For more information, refer above to “Selecting an Element during Parameter Entry.”)
- ☐ **Current Gain:** Must be a real number. The gain relates the source current to the controlling current by:

$$I(\text{Source}) = \text{Gain}[I(\text{Controller})],$$

where the direction of positive current flow is from the positive node through the controlling source to the negative node.

G-VCCS (Voltage-Controlled Current Source)

Upon selecting an element of this type, the following element parameter screen appears:

```

      G-VCCS
→ELEMENT: PRESS ENTER
  NODE (+): PRESS ENTER
  NODE (-): PRESS ENTER
CONTROL NODE (+): PRESS ENTER
CONTROL NODE (-): PRESS ENTER
TRANSCONDUCTANCE (MHO): PRESS ...

MAIN →STEP PRINT VIEW FONT UP
```

You should now enter or edit the parameter values. When you have finished, press **UP** to return to the Element Manager, and the element will be saved automatically.

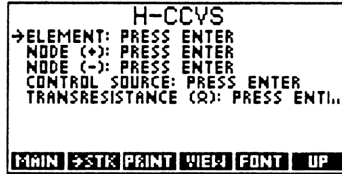
Here are specific descriptions of these parameters:

- ☐ **Element:** Name must begin with “G” and include no spaces. Possible names include: “G0,” “G74,” “Gnode7node4,” etc.
- ☐ **Node (+):** Selected from the Node Manager. (For more information, refer above to “Selecting a Node during Parameter Entry.”)
- ☐ **Node (-):** Selected from the Node Manager. The (+) and (-) nodes must be different.
- ☐ **Control Node (+):** Selected from the Node Manager.
- ☐ **Control Node (-):** Selected from the Node Manager. The control (+) and control (-) nodes must be different.
- ☐ **Transconductance (mho):** Must be a real number, will automatically have units of mhos (Siemens). The transconductance relates the source current to the controlling voltage by:

$$I(\text{Source}) = \text{Transcon.} [V(C(+)) - V(C(-))].$$

H-CCVS (Current-Controlled Voltage Source)

Upon selecting an element of this type, the following element parameter screen appears:



```
H-CCVS
→ELEMENT: PRESS ENTER
NODE (+): PRESS ENTER
NODE (-): PRESS ENTER
CONTROL SOURCE: PRESS ENTER
TRANSRESISTANCE (Ω): PRESS ENTI...

MAIN →STR PRINT VIEW FONT UP
```

You should now enter or edit the parameter values. When you have finished, press **UP** to return to the Element Manager, and the element will be saved automatically.

Here are specific descriptions of these parameters:

- ☐ **Element:** Name must begin with “H” and include no spaces. Possible names include: “H0,” “H74,” “Hnode7node4,” etc.
- ☐ **Node (+):** Selected from the Node Manager. (For more information, refer above to “Selecting a Node during Parameter Entry.”)
- ☐ **Node (-):** Selected from the Node Manager. The (+) and (-) nodes must be different.
- ☐ **Control Source:** Must be an independent voltage source. Selected from the Element Manager. (For more information, refer above to “Selecting an Element during Parameter Entry.”)
- ☐ **Transresistance (Ω):** Must be a real number, will automatically have units of ohms. The transresistance relates the source voltage to the controlling current by:

$$V(\text{Source}) = \text{Transres.} [I(\text{Controller})],$$

where the direction of positive current flow is from the positive node through the controlling source to the negative node.

Transient Sources

Independent voltage and current sources can be either constant or transient. Performing a Transient Sweep (TR) analysis of a circuit will automatically take into account transient variations of sources if they have been specified during element parameter entry. There is no limit on the number of sources that can be given transient variations, nor is there a requirement that all transients be of the same type.

To enter a transient variation for an independent source, go to the Element Editor and add or edit an independent voltage or current source. At the resulting element parameter screen, move the pointer to “Transient” and press **ENTER**. You will be prompted for the type of transient:

Pick Transient:			
NONE	PULSE	PWL	
FUNCTION			
MAIN			UP

Move the cursor to the desired transient type and press **ENTER**. (Instead of pressing **ENTER**, you can press **UP** to abort the operation, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.)

If you select a transient type of NONE, that will erase any previously existing transient for the current element. If you select any other transient type, you will be presented with a screen containing a number of parameters required to specify the chosen type of transient (the “transient parameter screen”). The exact parameters for each of the various types of transients are explained in detail below.

For tips on transient parameter entry, refer to the section above, “Element Parameter Entry Tips.”

Pulse Transient

Upon selecting a transient of this type, the following transient parameter screen appears:

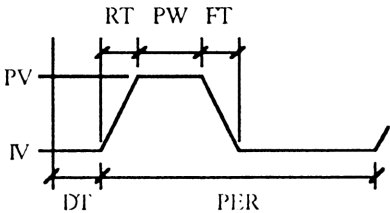
Pulse Transient

→INITIAL VALUE (V,A): PRESS ENTER
PULSED VALUE (V,A): PRESS ENTER
DELAY TIME (S): PRESS ENTER
RISE TIME (S): PRESS ENTER
FALL TIME (S): PRESS ENTER
PULSE WIDTH (S): PRESS ENTER
PERIOD (S): 9.6499

MAIN → SET PRINT VIEW FONT UP

You should now enter or edit the parameter values. When you have finished, press **UP** to return to the element parameter screen, and the transient will be converted to a user-defined function and saved automatically.

Here is a general diagram of a pulse transient (with abbreviated parameters):



Here are specific descriptions of these parameters:

- ☐ **Initial Value (V,A):** Initial and final value of the source, before the pulse rises and after the pulse falls. Must be a real number, will automatically have units of volts or amperes (depending on source type).
- ☐ **Pulsed Value (V,A):** Must be a real number, will automatically have units of volts or amperes (depending on source type).
- ☐ **Delay Time (s):** Time at which pulse starts rising. Must be a non-negative real number, will automatically have units of seconds.
- ☐ **Rise Time (s):** Time required for pulse to rise from initial value to pulsed value. Must be a positive real number, will automatically have units of seconds. A zero rise time is not allowed, but it can be made arbitrarily small.
- ☐ **Fall Time (s):** Time required for pulse to fall from pulsed value to initial value. Must be a positive real number, will automatically have

units of seconds. A zero fall time is not allowed, but it can be made arbitrarily small.

- ☐ **Pulse Width (s):** Duration of pulse after rising and before falling. Must be a positive real number, will automatically have units of seconds. A zero pulse width is not allowed, but it can be made arbitrarily small.
- ☐ **Period (s):** Time after which this pulse transient repeats. Must be a non-negative real number, will automatically have units of seconds, defaults to 9.E499. A non-periodic transient should have a value of 0 or 9.E499 for the period, as these values will have no effect.

PWL Transient

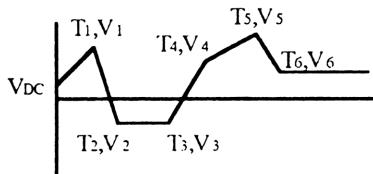
Upon selecting a transient of this type, the following transient parameter screen appears:

```
PWL Transient
→PWL LIST: PRESS ENTER
PERIOD (S): 9.E499

MAIN  →STEP PRINT VIEW FONT  UP
```

You should now enter or edit the parameter values. When you have finished, press **UP** to return to the element parameter screen, and the transient will be converted to a user-defined function and saved automatically.

Here is a general diagram of a PWL transient, where V_{DC} is the DC Value of the source:



Here are specific descriptions of these parameters:

- ☐ **PWL List:** Pairs of times and source values describing the piecewise-linear transient. Times: must be non-negative real numbers, must strictly (monotonically) increase with no duplications, will automatically have units of seconds. Source values: must be real numbers, will

automatically have units of volts or amperes (depending on source type). The entry sequence is: time1 value1 time2 value2 time3 value3 etc., with the times and values alternating. There must be an even number of arguments for a valid PWL list. If you specify a PWL transient for a source and enter a value for time zero in the PWL list, that value will override the DC Value.

- ☐ **Period (s):** Time after which this PWL transient repeats. Must be a non-negative real number, will automatically have units of seconds, defaults to 9.E499. A non-periodic transient should have a value of 0 or 9.E499 for the period, as these values will have no effect.

Function Transient

Upon selecting a transient of this type, the following transient parameter screen appears:

```

Function Transient
→FUNCTION: PRESS ENTER
PERIOD (S): 9.E499

MAIN →STK PRINT VIEW FONT UP
  
```

You should now enter or edit the parameter values. When you have finished, press **UP** to return to the element parameter screen, and the transient will be converted to a user-defined function and saved automatically.

Here are specific descriptions of these parameters:

- ☐ **Function:** An arbitrary algebraic function describing the transient. The function must be of only one variable, which can be anything, as the variable will automatically be converted to f to avoid conflict with existing global variables in your HP 48SX. The function *must* be surrounded by tic marks (') to indicate it is an algebraic.

NOTE: The trigonometric setting of your HP 48SX at the time you run Spice48 will affect trigonometric function transients.

- ☐ **Period (s):** Time after which this function transient repeats. Must be a non-negative real number, will automatically have units of seconds, defaults to 9.E499. A non-periodic transient should have a value of 0 or 9.E499 for the period, as these values will have no effect. If the specified function is periodic (e.g., $\sin(x)$), a period specified here will be imposed upon that function.

Chapter 5

Analysis Manager


The Analysis Manager is used to analyze the current circuit. (For more information on how to select the current circuit, refer to “Selecting the Current Circuit” in Chapter 3.)

This chapter covers:

- ☐ Using the Analysis Manager
- ☐ Preparing the Current Circuit
- ☐ Analysis Parameter Entry Tips
- ☐ Operating Point (OP) Analysis
- ☐ Transfer Function (TF) Analysis
- ☐ DC Sweep (DC) Analysis
- ☐ AC Sweep (AC) Analysis
- ☐ Transient Sweep (TR) Analysis

Using the Analysis Manager

To get to the Analysis Manager, follow these steps:

- ❶ Press  **LIBRARY** to display all libraries available to your HP 48SX.
- ❷ Find and press **SPICE** to enter the Spice48 Pac library directory.
- ❸ Press the first softkey, **SPICE**, to start the Spice48 Pac.
- ❹ At the Main menu, make sure the pointer is at “Analysis Manager” and press **ENTER**:



The Analysis Manager lists the five types of analysis that can be performed on the current circuit. (For more information on how to select the current circuit, refer to “Selecting the Current Circuit” in Chapter 3.) A form of analysis is selected by moving the pointer to the desired item and pressing **ENTER**.

Items in the Analysis Manager






Each item in the Analysis Manager is briefly described below and is discussed in detail in the various sections of this chapter.

Item	Description
Operating Point	Calculates DC or AC steady-state voltages and currents for the current circuit.
Transfer Function	Calculates DC or AC transfer function at any two nodes and input and output impedances for the current circuit.
DC Sweep	Calculates DC steady-state voltages and currents for the current circuit for a series of values (sweep) of an independent voltage or current source.
AC Sweep	Calculates AC steady-state voltages and currents for the current circuit for a linear, octave, or decade series (sweep) of frequencies.
Transient Sweep	Calculates the transient response of the current circuit based on DC or transient values of sources and optional initial conditions on circuit elements.

Summary of Operations

Screen	Softkeys
Analysis Manager	MAIN -STK PRINT VIEW FONT UP

Key	Action
FONT	Toggles between the small and large fonts.
MAIN	Returns to the Main menu.
PRINT	Prompts for ONE or ALL to select items, and then sends those items to an IR printer.
-STK	Prompts for ONE or ALL to select items, and then copies those items to the stack. The items are placed in a list if ALL was chosen.

	Moves up one level in the menu structure, returning to the Main menu.
	Displays the entire text of an item too wide to fit on the screen, up to one entire screen size. If the item fits on the screen, this key is non-functional.
	Quits the Spice48 Pac to the HP 48SX stack.
	Moves down one level in the menu structure, entering the analysis parameter screen.
	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Preparing the Current Circuit

After you have selected a type of analysis, you will be presented with a screen containing a number of parameters required to specify the chosen type of analysis (the “analysis parameter screen”). But first, if any of the following conditions are true, there will be a delay while the circuit is prepared:

- ☐ This is the first analysis to take place on the current circuit.
- ☐ The current circuit has been packed since the last preparation.
- ☐ Elements have been added, deleted, or edited since the last preparation.
- ☐ Nodes have been added since the last preparation.
- ☐ The reference node has been changed since the last preparation.

The preparation delay will be short for small circuits with only a few nodes, but may be several minutes for circuits with ten or more nodes.

Circuit preparation involves the following:

- ☐ Unused nodes are deleted.
- ☐ Extra nodes are added to track currents in voltage sources and inductors. When viewing data with the Output Manager, you will be able to see voltages at circuit nodes and currents at extra nodes.
- ☐ Transients of independent sources are organized.

- ☐ Each node is checked for connections to at least two elements.
- ☐ Matrices are created that describe the current circuit
- ☐ Column vectors are created that describe the sources.
- ☐ TR analysis integration parameters are deleted because they have now become obsolete. (For more information, refer below to “Transient Sweep (TR).”)

All of the global variables created at this time are stored into the current circuit’s subdirectory of the ‘SPICE48D’ directory. Later, if you run low on free memory, you can pack the current circuit, which will remove these global variables. (For more information, refer to “Memory Requirements” in Chapter 1 and “Packing the Current Circuit” in Chapter 3.)

After the current circuit has been successfully prepared (or if it was already prepared), you will be presented with the analysis parameter screen. The exact parameters for each of the various types of analysis are explained in detail in the sections below.

Analysis Parameter Entry Tips

Here are some useful things to know about entering or editing analysis parameters:

- ☐ To enter or edit the value of a parameter, move the pointer to it and press **ENTER**. After entering or editing the value of a parameter, press **ENTER** to accept the new value and return to the analysis parameter screen.
- ☐ Once you have begun entering or editing a parameter value, to abort the change, press **ATTN** to clear the command line (if necessary) and then press **ENTER** or **ATTN** to return to the analysis parameter screen without changing the parameter value.
- ☐ You only need to enter values for those parameters that initially display, “Press ENTER.” Other parameters (e.g., the frequency for an OP analysis, the calculation speed for a TR analysis) will be given default values (e.g., 0 and FAST, respectively) which you may modify if you wish.
- ☐ When you are entering a numeric value, make use of the softkeys to append the desired exponent to the value (e.g., type 3 and then press **MICRO** to enter the value .000003). More exponent softkeys can be accessed by pressing **NXT**.

- ❑ Do not be surprised if a numeric value you enter for an analysis parameter *changes* between the moment you press **ENTER** to accept it and the moment it is displayed on the analysis parameter screen. This automatic adjustment is necessary to guarantee that all analysis parameters are valid. (For more information, refer below to the appropriate section.)
- ❑ It is possible that after specifying values for a series of related analysis parameters (e.g., start, stop, and step for a DC analysis), you are not allowed to change any of them because doing so would create inconsistent values. If this happens, press **UP** to return to the Analysis Manager and then press **ENTER** to immediately re-enter the analysis parameter screen. This will reset the analysis parameters to their default values. (For more information, refer below to the appropriate section.)
- ❑ An OP analysis is faster than a DC or AC analysis if you only want the steady-state solution for *one* value of a source. However, you *are* allowed to sweep a source from X to X with a step of 0.

Selecting a Node during Parameter Entry

To specify a TF analysis, you must select both output (+) and output (–) nodes. When you press **ENTER** to select the (+) or (–) node of the TF analysis, you will be presented with a list of all of the nodes in the current circuit. To pick one of the nodes, move the cursor to the desired node and press **ENTER**. (Instead of pressing **ENTER**, you can press **UP** to abort the operation, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.) If you are *changing* the (+) or (–) node of the TF analysis, the cursor will automatically be placed on the current node; otherwise, the cursor will be placed on the first node in the list.

You are not allowed to add nodes to the current circuit at this time because the circuit has been prepared. (For more information on adding nodes, refer to Chapter 6, “Node Manager.”)

Selecting an Element during Parameter Entry

To specify a DC or AC analysis, you must select an independent source to sweep. When you press **ENTER** to select the source to vary, you will be presented with a list of all of the elements in the current circuit. To pick one of the elements, move the cursor to the desired element and press **ENTER**. (Instead of pressing **ENTER**, you can press **UP** to abort the operation, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.) You will only be allowed to select an appropriate type of element (e.g., an independent voltage or current source). If you are *changing* the independent source of the DC or AC analysis, the cursor will automatically be placed on the current independent source; otherwise, the cursor will be placed on the first element in the list.

You are not allowed to add elements to the current circuit at this time because the circuit has been prepared. (For more information on adding elements, refer to Chapter 4, “Element Manager.”)

Summary of Operations

Screen	Softkeys
Analysis Parameter	MAIN -STK PRINT CALC FONT UP

Key	Action
CALC	Begins an analysis, using the specified analysis parameters.
FONT	Toggles between the small and large fonts.
MAIN	Returns to the Main menu.
PRINT	Prompts for ONE or ALL to select items, and then sends those items to an IR printer.
-STK	Prompts for ONE or ALL to select items, and then copies those items to the stack. The items are placed in a list if ALL was chosen.
UP	Moves up one level in the menu structure, returning to the Analysis Manager.
ATTN	Quits the Spice48 Pac to the HP 48SX stack. The element is saved automatically.
ENTER	Prompts for the value of the parameter currently selected by the pointer.
ON PREV	Dumps the current screen to an IR printer.
VIEW	Displays the entire text of an item too wide to fit on the screen, up to one entire screen size. If the item fits on the screen, this key is non-functional.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Operating Point (OP) Analysis

An OP analysis on the current circuit calculates the steady-state voltage at each node and the steady-state current at each voltage source or inductor. DC values are returned if the frequency is zero, or AC values if the frequency is non-zero.

Upon selecting an analysis of this type, the following analysis parameter screen appears:



You should now enter or edit the parameter values. When you have finished, press **CALC** to perform the actual calculation. The results of the calculation will be displayed in the Output Manager Data View. (For more information, refer to Chapter 9, “Output Manager.”)

Here are specific descriptions of these parameters:

- ☐ **Frequency (Hz):** Must be a non-negative real number, will automatically have units of Hertz, defaults to 0. Calculated values will be DC values if frequency is zero, AC values if non-zero.
- ☐ **Memory:** An estimate of the free memory required to perform the analysis. You cannot modify this parameter. (For more information, refer to “Memory Requirements” in Chapter 1.)

Transfer Function (TF) Analysis

A TF analysis on the current circuit calculates the input impedance “seen by” the source, the output impedance between *any* two nodes (neither of which is required to be the reference node), and the gain from the input to the output. DC values are returned if the frequency is zero, or AC values if the frequency is non-zero.

Upon selecting an analysis of this type, the following analysis parameter screen appears:



You should now enter or edit the parameter values. When you have finished, press **CALC** to perform the actual calculation. The results of the calculation will be displayed in the Output Manager Data View. (For more information, refer to Chapter 9, “Output Manager.”)

Here are specific descriptions of these parameters:

- ☐ **Source:** Must be an independent voltage or current source. Selected from the Element Manager. (For more information, refer above to “Selecting an Element during Parameter Entry.”)
- ☐ **Output Node (+):** Selected from the Node Manager. (For more information, refer above to “Selecting a Node during Parameter Entry.”)
- ☐ **Output Node (-):** Defaults to the reference node. Selected from the Node Manager. The output (+) and output (-) nodes must be different.
- ☐ **Frequency (Hz):** Must be a non-negative real number, will automatically have units of Hertz, defaults to 0. Calculated values will be DC values if frequency is zero, AC values if non-zero.
- ☐ **Memory:** An estimate of the free memory required to perform the analysis. You cannot modify this parameter. (For more information, refer to “Memory Requirements” in Chapter 1.)

DC Sweep (DC) Analysis

A DC analysis on the current circuit calculates the steady-state voltage at each node and the steady-state current at each voltage source or inductor for a series of values (a sweep) of an independent voltage or current source. The frequency is always zero for a DC analysis, so DC values are returned.

Upon selecting an analysis of this type, the following analysis parameter screen appears:



You should now enter or edit the parameter values. When you have finished, press **CALC** to perform the actual calculation. The results of the calculation will be displayed in the Output Manager Node View. (For more information, refer to Chapter 9, “Output Manager.”)

Here are specific descriptions of these parameters:

- ☐ **Source:** Must be an independent voltage or current source. Selected from the Element Manager. (For more information, refer above to “Selecting an Element during Parameter Entry.”)
- ☐ **Start (V,A):** Must be a real number, will automatically have units of volts or amperes (depending on source type). The start value must not be greater than the stop value.
- ☐ **Stop (V,A):** Must be a real number, will automatically have units of volts or amperes (depending on source type). The stop value must not be less than the start value. The stop value will automatically be adjusted upward so that an integral number of steps occur between the start and stop values.
- ☐ **Step (V,A):** Must be a non-negative real number, will automatically have units of volts or amperes (depending on source type). There must be an integral number of steps between the start and stop values.
- ☐ **Memory:** An estimate of the free memory required to perform the analysis. You cannot modify this parameter. (For more information, refer to “Memory Requirements” in Chapter 1.)

AC Sweep (AC) Analysis

An AC analysis on the current circuit calculates the steady-state voltage at each node and the steady-state current at each voltage source or inductor for a series (a sweep) of frequencies. The frequency is never zero for an AC analysis, so AC values are returned.

Upon selecting an analysis of this type, the following analysis parameter screen appears:



You should now enter or edit the parameter values. When you have finished, press **CALC** to perform the actual calculation. The results of the calculation will be displayed in the Output Manager Node View. (For more information, refer to Chapter 9, “Output Manager.”)

Here are specific descriptions of these parameters:

- ☐ **Source:** Must be an independent voltage or current source. Selected from the Element Manager. (For more information, refer above to “Selecting an Element during Parameter Entry.”)
- ☐ **Start (Hz):** Must be a positive real number, will automatically have units of Hz. The start value must not be greater than the stop value.
- ☐ **Stop (Hz):** Must be a positive real number, will automatically have units of Hz. The stop value must not be less than the start value.
- ☐ **No. Points:** Must be a positive integer. For the DECADE and OCTAVE output scales, this parameter represents the number of points *per* decade or octave; for the LINEAR output scale, this parameter represents the *total* number of points.
- ☐ **Output Scale:** Selected from DECADE, LINEAR, and OCTAVE, defaults to DECADE. Decade and octave output scales will automatically invoke semi-logarithmic plotting.

To change the output scale at the analysis parameter screen, move the pointer to “Output Scale” and press **ENTER**. Then, at the next screen, move the cursor to the desired output scale and press **ENTER**. (Instead of pressing **ENTER**, you can press **UP** to abort the operation, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.)

- ☐ **Memory:** An estimate of the free memory required to perform the analysis. You cannot modify this parameter. (For more information, refer to “Memory Requirements” in Chapter 1.)

Transient Sweep (TR) Analysis

A TR analysis on the current circuit calculates the transient response. You can choose to have the initial values on the elements be those you specified during element entry or be from a steady-state OP analysis of the current circuit. Transient variations of sources specified during element parameter entry will automatically be taken into account, with no limit on the number of sources with transient variations. (For more information, refer to “Transient Sources” in Chapter 4.)

Upon selecting an analysis of this type, one of two screens will appear, based on one of the following:

- ☐ **New TR Analysis:** If this is the first TR analysis to be performed on the current circuit, or if the current circuit has been re-prepared since the last TR analysis, then integration parameters do not exist, and you should continue below with “New TR Analysis.” (For more information on circuit preparation, refer above to “Preparing the Current Circuit.”)
- ☐ **Continuing TR Analysis:** If a previous TR analysis has been performed on the current circuit and the current circuit has not been re-prepared, then integration parameters do exist, and you should skip below to “Continuing TR Analysis.” (For more information on circuit preparation, refer above to “Preparing the Current Circuit.”)

New TR Analysis

The following analysis parameter screen appears:



```

      Transient Sweep
→SAVE START (S): PRESS ENTER
SAVE STOP (S): PRESS ENTER
SAVE STEP (S): .000125
CALC. STEP (S): .000125
USE OP OR IC: IC
CALC. SPEED: FAST
MEMORY: ???/15504.5
MAIN <STEP PRINT CALC FONT UP

```

You should now enter or edit the parameter values. When you have finished, press **CALC** to perform the actual calculation. The results of the calculation will be displayed in the Output Manager Node View. (For more information, refer to Chapter 9, “Output Manager.”)

Here are specific descriptions of these parameters:

- ☐ **Save Start (s):** Must be a non-negative real number, will automatically have units of seconds. The save start value must not be greater than the save stop value.

- ☐ **Save Stop (s):** Must be a non-negative real number, will automatically have units of seconds. The save stop value must not be less than the save start value. The save stop value will automatically be adjusted upward so that an integral number of save steps occur between the save start and save stop values.
- ☐ **Save Step (s):** Must be a non-negative real number, will automatically have units of seconds, defaults to the suggested calculation step. There must be an integral number of save steps between the save start and save stop values.
- ☐ **Calc Step (s):** Must be a non-negative real number, will automatically have units of seconds, defaults to a suggested value. There must be an integral number of calculation steps per save step. The calculation step value will automatically be adjusted downward so that an integral number of calculation steps occur every save step.
- ☐ **Use OP or IC:** Selected from OP and IC, defaults to IC. This parameter represents the type of initial conditions to use at time zero of the TR analysis. If IC is selected, the initial values of the capacitors and inductors will be used. If OP is selected, the initial values will be ignored and an OP analysis with zero frequency will be performed, with the resulting voltages and currents being used.

To change the initial conditions type at the analysis parameter screen, move the pointer to “Use OP or IC” and press **ENTER**. Then, at the next screen, move the cursor to the desired initial conditions type and press **ENTER**. (Instead of pressing **ENTER**, you can press **UP** to abort the operation, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.)

- ☐ **Calc Speed:** Selected from FAST, COMPROMISE, and ACCURATE, defaults to FAST. This parameter represents a tradeoff between accuracy and speed, with ACCURATE taking the most time but returning the most accurate results. The extra delay required for COMPROMISE or ACCURATE only occurs during the TR integration setup. (For more information, refer to Appendix C, “Technical Information.”)

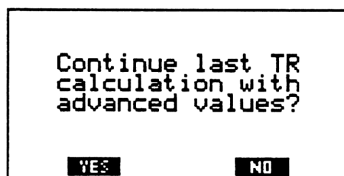
To change the calculation speed at the analysis parameter screen, move the pointer to “Calc Speed” and press **ENTER**. Then, at the next screen, move the cursor to the desired calculation speed and press **ENTER**. (Instead of pressing **ENTER**, you can press **UP** to abort the operation, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.)

- ☐ **Memory:** An estimate of the free memory required to perform the analysis. You cannot modify this parameter. (For more information, refer to “Memory Requirements” in Chapter 1.)

Continuing TR Analysis

A TR analysis is quite complex and requires the preparation of a number of integration parameters and matrices *in addition* to the usual circuit preparation described above in “Preparing the Current Circuit.” Also, you may be interested in the transient response of the current circuit over an extended period of time involving possibly *hundreds* of time values, many more than might be calculated at any one time due to insufficient free memory in your HP 48SX. (Users with merged RAM cards are unlikely to encounter this situation.)

For these reasons, there is an automatic continuation feature to the TR analysis. After you perform a TR analysis of the current circuit, if you later return to the Analysis Manager and again select a TR analysis, the following screen appears:



Press **YES** to automatically continue the last TR analysis (described below), press **NO** to display the analysis parameter screen with the current values of the analysis parameters for editing (described above in “TR Analysis (New)”), or press **ATTN** to abort the operation and return to the Analysis Manager.

An automatically continued TR analysis will advance the time values as demonstrated in this example:

- ① Analysis parameters for the very first TR analysis of the current circuit:
 - ☐ **Begin:** TR analysis begins at time 0 seconds.
 - ☐ **Save Start:** 4 seconds.
 - ☐ **Save Stop:** 10 seconds.
 - ☐ **Save Step:** 1 second.
 - ☐ **Values:** Returned for times 4, 5, 6, 7, 8, 9, 10 seconds.
- ② Analysis parameters for an automatically continued TR analysis of the current circuit:
 - ☐ **Begin:** TR analysis begins at time 10 seconds.
 - ☐ **Save Start:** 14 seconds (incremented by length of last TR analysis).
 - ☐ **Save Stop:** 20 seconds (incremented by length of last TR analysis).
 - ☐ **Save Step:** 1 second (remains unchanged).
 - ☐ **Values:** Returned for times 14, 15, 16, 17, 18, 19, 20 seconds.
- ③ And so forth for all further continuations of this TR analysis.

Chapter 6

Node Manager



The Node Manager is used to add, delete, and edit nodes in the current circuit and to set the reference node. (For more information on how to select the current circuit, refer to “Selecting the Current Circuit” in Chapter 3.)

This chapter covers:

- ☐ Using the Node Manager
- ☐ Adding a Node
- ☐ Deleting a Node
- ☐ Editing a Node
- ☐ Setting the Reference Node

Using the Node Manager

To get to the Node Manager, follow these steps:

- ❶ Press  **LIBRARY** to display all libraries available to your HP 48SX.
- ❷ Find and press **SPICE** to enter the Spice48 Pac library directory.
- ❸ Press the first softkey, **SPICE**, to start the Spice48 Pac.
- ❹ At the Main menu, make sure the pointer is at “Node Manager” and press .

Items in the Node Manager

The items listed in the Node Manager are the nodes in the current circuit. The following list of nodes is from tutorial circuit BJTDC (For more information, refer to Chapter 2, “Spice48 Tutorial.”):

Nodes:		
SOURCE	(REFER) GR	INPUT
BASE	BATTERY	EMITTER
MMETER	COLLECTOR	OUTPUT
ADD	DEL	EDIT REFER
		UP

If the current circuit has no nodes, the screen will display, “Press ADD.”

Summary of Operations

Screen	Softkeys
Node Manager	ADD DEL EDIT REFER UP

Key	Action
ADD	Adds one or more new nodes to the current circuit.
DEL	Deletes the node currently selected by the cursor.
EDIT	Edits the node currently selected by the cursor.
REFER	Sets the node currently selected by the cursor to be the reference node for the current circuit.
UP	Moves up one level in the menu structure, returning to the Main menu.
ATTN	Quits the Spice48 Pac to the HP 48SX stack.
ENTER	Edits the node currently selected by the cursor, or if no nodes exist, adds one or more new nodes to the current circuit
ON—PREV	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Adding a Node

To add one or more new nodes, go to the Node Manager and press **ADD**. You will be prompted for the name(s) of the node(s) to add:



The screenshot shows a terminal window titled "PRG" with the text "{ HOME SPARCOM SPICE4BD BJTDC }". Below this, it says "Enter node name(s):" followed by "→ node1 node2 ..." and "< names >". At the bottom, there is a row of softkeys: "←SKIP SKIP→", "←DEL DEL→", "INS", and "↑STK".

Type in as many new node names as you wish (separated by spaces) and press **ENTER**, or to abort the add operation, press **ATTN** to clear the command line (if necessary), and then press **ENTER** or **ATTN** to return to the Node Manager without adding any new nodes.. (For more information on how to use the editing softkeys, refer to “Text Editing” in Chapter 1.)

Deleting a Node

To delete a node, go to the Node Manager, move the cursor to the desired node, and press **DEL**. You will be prompted to verify the delete operation by pressing **YES** to confirm or **NO** or **ATTN** to abort. If you confirm the delete operation, the current circuit will be searched for references to the node, and if no references are found, the node will be deleted. If the node is referenced by any elements in the current circuit, you will first have to either delete those elements or else change the node references to some other node.

When you pack or analyze the current circuit, all unused nodes are automatically deleted, so you do not need to worry if you think you have added extra nodes to the current circuit.

Editing a Node

To edit a node, go to the Node Manager, move the cursor to the desired node, and press **EDIT**. You will be prompted to edit the node name. Press **ENTER** to accept changes to the node name, or to abort the edit operation, press **ATTN** to clear the command line (if necessary) and then press **ENTER** or **ATTN** to return to the Node Manager without editing the node. (For more information on how to use the editing softkeys, refer to “Text Editing” in Chapter 1.)

Setting the Reference Node

The reference node is used by Spice48 as the circuit ground for all calculations. By default, the reference node is set to the (–) node of the first element entered into the current circuit. (Typically, the first element you enter will be an independent source connected to ground.) However, it may be the case that this is incorrect, and you should reset the reference node to the proper ground node.

To set the reference node, go to the Node Manager, move the cursor to the desired node, and press **REFER**. This will set the node currently selected by the cursor to be the reference node, and that node will be indicated by the label “(REFER)” in the Node Manager.

Chapter 7

Graphics Manager


The Graphics Manager is used to create the schematic of the current circuit. (For more information on how to select the current circuit, refer to “Selecting the Current Circuit” in Chapter 3.)

This chapter covers:

- ☐ Using the Graphics Manager
- ☐ Creating an Accurate Schematic
- ☐ Coordinate Entry Tips
- ☐ Setting Node Coordinates
- ☐ Setting Element Coordinates and Orientations
- ☐ Setting Wire Coordinates
- ☐ Viewing and Printing the Schematic
- ☐ Clearing All Graphics

Using the Graphics Manager

To get to the Graphics Manager, follow these steps:

- ❶ Press  **LIBRARY** to display all libraries available to your HP 48SX.
- ❷ Find and press **SPICE** to enter the Spice48 Pac library directory.
- ❸ Press the first softkey, **SPICE**, to start the Spice48 Pac.
- ❹ At the Main menu, make sure the pointer is at “Graphics Manager” and press **ENTER**:



The Graphics Manager lists the various graphics functions necessary to create the schematic of the current circuit. (For more information on how to select the current circuit, refer to “Selecting the Current Circuit” in Chapter 3.) A function is selected by moving the pointer to the desired item and pressing **ENTER**.

Items in the Graphics Manager

Each item in the Graphics Manager is briefly described below and is discussed in detail in the various sections of this chapter.

Item	Description
Set Node Coordinates	Edits node coordinates.
Set Elmt Coordinates	Edits element coordinates or automatically generates them.
Set Wire Coordinates	Adds, deletes, and edits wire coordinates or automatically generates them.
View Schematic	Displays the schematic.
Print Schematic	Prints the schematic.
Clear All Graphics	Deletes all coordinates and the schematic.

Summary of Operations

Screen	Softkeys
Graphics Manager	MAIN →STK PRINT VIEW FONT UP

Key	Action
FONT	Toggles between the small and large fonts.
MAIN	Returns to the Main menu.
PRINT	Prompts for ONE or ALL to select items, and then sends those items to an IR printer.
→STK	Prompts for ONE or ALL to select items, and then copies those items to the stack. The items are placed in a list if ALL was chosen.
UP	Moves up one level in the menu structure, returning to the Main menu.

VIEW	Displays the entire text of an item too wide to fit on the screen, up to one entire screen size. If the item fits on the screen, this key is non-functional.
ATTN	Quits the Spice48 Pac to the HP 48SX stack.
ENTER	Moves down one level in the menu structure, entering the selected function.
ON PREV	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Creating an Accurate Schematic

Here are some useful things to know about creating circuit schematics that will help you to create accurate schematics:

- ☐ You must always specify all of the node coordinates. From the node coordinates, Spice48 can automatically generate the locations of all the circuit elements and create wires to connect them where necessary, but it cannot initially place the nodes.
- ☐ Coordinates of nodes, elements, and wires fall on an imaginary grid with integer grid coordinates. Although there is no restriction on the coordinates you enter, it will be easiest to imagine and describe your circuit if you start with ground (the reference node) at the origin (0,0) and locate the remaining nodes from there. Both negative and positive coordinates are allowed, so if you do place the reference node at (0,0), you are free to place the remaining nodes in any direction. Horizontally, right is positive; vertically, up is positive.
- ☐ Nodes should typically be spaced a single unit apart—e.g., (0,0), (1,0), and (1,1) for a simple circuit. Each unit in the imaginary circuit schematic grid corresponds to 20 pixels on your HP 48SX screen, so if you place the nodes more than one unit apart, the schematic may end up larger than the screen. (For complex circuits, this may be unavoidable.) The smaller the schematic, the less free memory required to create, view, and save it. Schematics larger than the screen will be scrollable.
- ☐ In general, follow these steps to create an accurate schematic:
 - ① Set all of the node coordinates. (For more information, refer below to “Setting Node Coordinates.”)

- ② Automatically generate the element coordinates. (For more information, refer below to “Setting Element Coordinates and Orientations.”)
- ③ For complex schematics—specifically, those with elements in parallel—the elements may overlap or need to be relocated or re-oriented. View the schematic to check the automatically generated element locations and orientations. Make a note of the elements requiring adjustment, and then exit the schematic view and adjust the coordinates and/or orientations of those elements to clean up the schematic.
- ④ Automatically generate the wire coordinates. (For more information, refer below to “Setting Wire Coordinates.”)
- ⑤ For some schematics, you may want diagonal wires, but automatically generated wires will always be horizontal or vertical. If this is the case, view the schematic to check the automatically generated wire locations. Make a note of the wires requiring adjustment or removal, and then exit the schematic view and add, delete, or edit the wires as required to optimize the schematic.
- ⑥ View or print the finished schematic. (For more information, refer below to “Viewing and Printing the Schematic.”)

Coordinate Entry Tips

Here are some useful things to know about entering and editing coordinates of nodes, elements, and wires:

- ☐ To enter or edit the coordinates of a node, element, or wire, move the cursor to it and press **EDIT** or **ENTER**. After entering or editing the coordinates, press **ENTER** to accept the new coordinates and return to the appropriate coordinates screen.
- ☐ Once you have begun entering or editing coordinates of a node, element, or wire, to abort the change, press **ATTN** to clear the command line (if necessary) and then press **ENTER** or **ATTN** to return to the appropriate coordinates screen without changing the coordinates.

Setting Node Coordinates

You must always specify all of the node coordinates.

Upon selecting this function, the nodes in the current circuit will appear, with their coordinates. The following list of node coordinates is from tutorial circuit BJTDC (For more information, refer to Chapter 2, “Spice48 Tutorial.”):

Nodes:					
(0,1)	SOURC	(0,0)	GRD0	(0,2)	INPUT
(1,2)	BASE	(6,3)	EMITE	(2,1)	EMITT
(3,2)	WEME	(4,2)	COLL	(5,2)	OUTPU
MAIN		EDIT		CLEAR UP	

You should now edit the node coordinates. To edit the coordinates of a node, move the cursor to the desired node and press **ENTER** or **EDIT**. You will be prompted to enter or edit the node coordinates:

PRG	
{ HOME SPARCOM SPICE48D BJTDC }	
Edit coordinates:	
→ x y	
{ integers }	
0 1	
←SKIP SKIP→ ←DEL DEL→ INS ←STX	

Node coordinates consist of integer x and y locations in the imaginary circuit schematic grid. (For more information on the grid, refer above to “Creating an Accurate Schematic.”) When you have entered or edited the node coordinates, press **ENTER** to accept the new coordinates and return to the node screen, or to abort the operation, press **ATTN** to clear the command line (if necessary), and then press **ENTER** or **ATTN** to return to the node screen without changing the node coordinates. (For information on using the editing softkeys, refer to “Text Editing” in Chapter 1.)

Press **CLEAR** to delete all the current node coordinates.

When you have finished setting the node coordinates, press **UP** to return to the Graphics Manager, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.

Summary of Operations

Screen	Softkeys			
Node Coordinates	MAIN	EDIT	CLEAR	UP

Key	Action
CLEAR	Clears all of the current node coordinates.
EDIT	Prompts to enter or edit the coordinates of the node currently selected by the cursor.
MAIN	Returns to the Main menu.
UP	Moves up one level in the menu structure, returning to the Graphics Manager.
ATTN	Quits the Spice48 Pac to the HP 48SX stack.
ENTER	Prompts to enter or edit the coordinates of the node currently selected by the cursor.
ON — PREV	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Setting Element Coordinates and Orientations

You may either manually enter the element coordinates and orientations or automatically generate them from the node coordinates. If you choose to automatically generate them, you must first enter *all* of the node coordinates.

Upon selecting this function, the elements in the current circuit will appear, with their coordinates and orientations. The following list of element coordinates and orientations is from tutorial circuit BJTDC (For more information, refer to Chapter 2, “Spice48 Tutorial.”):

Elements:		
(0.1).1 VS	(0.1).5 RS	(0.2).7 CS
(1.3).1 R81	(1.2).1 R82	(1.2).8 V8E
(2.1).1 RE	(3.1).1 CE	(3.2).2 FCE
(4.2).3 V8M	(4.3).1 RC	(5.2).1 RL
(4.2).7 CL	(6.3).1 V8R	
MAIN	EDIT	AUTO CLEAR UP

You should now edit the element coordinates and orientations. To edit the coordinates or orientation of an element, move the cursor to the desired element and press **ENTER** or **EDIT**. You will be prompted to enter or edit the element coordinates and orientation:

```

PRG
{ HOME SPARCOM SPICE48D BJTDC }
Edit coordinates 456
and orientation: 3+7
→ x y orient(1-8) 218
0 1 1+
←SKIP SKIP→ ←DEL DEL→ INS ▢ ↑STK

```

Element coordinates consist of integer x and y locations in the imaginary circuit schematic grid. This location is the location of the element (+) node. (For more information on the grid, refer above to “Creating an Accurate Schematic.”)

Element orientations consist of an integer from 1 to 8 indicating the direction of the element (–) node from the element (+) node, as indicated at the prompt. (The + indicates the location of the element (+) node, while the surrounding integers indicate the direction of the (–) node from the (+) node.)

When you have entered or edited the element coordinates and orientation, press **ENTER** to accept the new coordinates and orientation and return to the element screen, or to abort the operation, press **ATTN** to clear the command line (if necessary), and then press **ENTER** or **ATTN** to return to the element screen without changing the element coordinates or orientation. (For information on using the editing softkeys, refer to “Text Editing” in Chapter 1.)

Press **AUTO** to automatically generate all the element coordinates and orientations from the node coordinates. This operation will replace all current element coordinates and orientations.

Press **CLEAR** to delete all the current element coordinates and orientations.

When you have finished setting the element coordinates and orientations, press **UP** to return to the Graphics Manager, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.

Summary of Operations

Screen	Softkeys
Element Coordinates	MAIN EDIT AUTO CLEAR UP

Key	Action
AUTO	Automatically generates element coordinates and orientations from the node coordinates.
CLEAR	Clears all of the current element coordinates and orientations.

EDIT	Prompts to enter or edit the coordinates and orientation of the element currently selected by the cursor.
MAIN	Returns to the Main menu.
UP	Moves up one level in the menu structure, returning to the Graphics Manager.
ATTN	Quits the Spice48 Pac to the HP 48SX stack.
ENTER	Prompts to enter or edit the coordinates and orientation of the element currently selected by the cursor.
ON PREV	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Setting Wire Coordinates

Wires are used to fix “loose” connections in a schematic, such as may happen when multiple elements *connect* to the same node but cannot directly *touch* that node. (This is nearly always the case for the reference node, which will connect to many elements but only be spatially located next to a single element.)

You may either manually enter wires or automatically generate them from the node coordinates and element coordinates and orientations. If you choose to automatically generate them, you must first enter *all* of the node coordinates and some or all of the element coordinates and orientations.

Upon selecting this function, the wires will appear. The following list of wires is from tutorial circuit BJTDC (For more information, refer to Chapter 2, “Spice48 Tutorial.”):

Wires:					
(0.0)	(6.0)	(0.0)	(1.0)	(0.0)	(5.0)
(0.0)	(3.0)	(0.0)	(2.0)	(1.3)	(6.3)
(1.1)	(1.0)	(2.1)	(3.1)	(4.3)	(6.3)
(5.1)	(5.0)	(6.2)	(6.0)		
ADD	DEL	EDIT	AUTO	CLEAR	UP

You should now add, delete, or edit the wires. To add a wire, press **ADD**. To delete a wire, move the cursor to the desired wire and press **DEL**. To edit a wire, move the cursor to the desired wire and press **ENTER** or **EDIT**. For adding and editing, you will be prompted to enter or edit the coordinates of the selected wire:


```

PRG
{ HOME SPARCOM SPICE48D BJTDC }
Edit coordinates:
→ x1 y1 x2 y2
( integers )
0 0 6 0
←SKIP SKIP→ ←DEL DEL→ INS ←↑STK

```

Wires consist of beginning and ending coordinates, each of which consist of integer x and y locations in the imaginary circuit schematic grid. (For more information on the grid, refer above to “Creating an Accurate Schematic.”) After you enter or edit the two coordinates describing a wire, the coordinates may be swapped to ensure that the left-most coordinate is listed first. Also, for your convenience, wires are automatically sorted by the left-most x location when listed at the wire screen.

When you have entered or edited the wire coordinates, press **ENTER** to accept the new coordinates and return to the wire screen, or to abort the operation, press **ATTN** to clear the command line (if necessary), and then press **ENTER** or **ATTN** to return to the wire screen without changing the wire coordinates. (For information on using the editing softkeys, refer to “Text Editing” in Chapter 1.)

Press **AUTO** to automatically generate all the wire coordinates from the node coordinates and element coordinates and orientations. This operation will replace all current wire coordinates.

Press **CLEAR** to delete all the current wire coordinates.

When you have finished setting the wire coordinates, press **UP** to return to the Graphics Manager or **ATTN** to quit Spice48.

Summary of Operations

Screen	Softkeys
Wire Coordinates	ADD DEL EDIT AUTO CLEAR UP

Key	Action
ADD	Prompts to enter the coordinates of a new wire and adds it.
AUTO	Automatically generates wire coordinates from the node coordinates and element coordinates and orientations.
CLEAR	Clears all of the current wire coordinates.
DEL	Deletes the wire currently selected by the cursor.

EDIT	Prompts to edit the coordinates of the wire currently selected by the cursor.
UP	Moves up one level in the menu structure, returning to the Graphics Manager.
ATTN	Quits the Spice48 Pac to the HP 48SX stack.
ENTER	Prompts to edit the coordinates of the wire currently selected by the cursor, or if no wires exist, prompts to enter the coordinates of a new wire and adds it.
ON — PREV	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

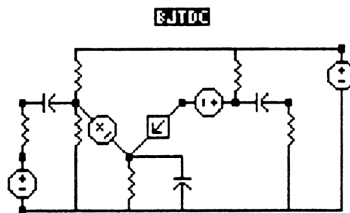
Viewing and Printing the Schematic

These functions allow you to view and print the schematic for the current circuit.

Viewing the Schematic

Upon selecting this function, the schematic for the current circuit will be created or updated (if necessary) and displayed.

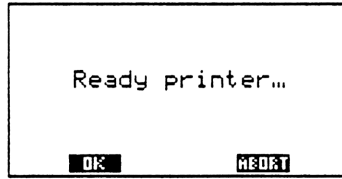
You will see the schematic as it is being built on the screen. When the schematic is completed, it will be saved and then re-displayed for viewing. If the schematic is larger than the screen, it will be automatically centered on the screen, and the cursor keys will be activated for scrolling. The following schematic is from tutorial circuit BJTDC (For more information, refer to Chapter 2, “Spice48 Tutorial.”):



When you have finished viewing the schematic, press **ENTER** or **ATTN** to return to the Graphics Manager.

Printing the Schematic

Upon selecting this function, the schematic for the current circuit will be created or updated (if necessary) and you will be prompted to ready the printer:



When you have turned on your IR printer and aligned your HP 48SX with it, press **OK**, or to abort the operation, press **ABORT** or **ATTN** to return to the Graphics Manager without printing the schematic.

Oversize schematics will automatically be printed in multiple sections by your IR printer.

Updating a Schematic

If an existing schematic is out-of-date because you have changed any of the node, element, or wire coordinates since it was last created or updated, before it will be displayed or printed, you will be prompted if you wish to update it:



Press **YES** to update the schematic and then view or print it, press **NO** to view or print the old version of the schematic, or press **ATTN** to abort the operation and return to the Graphics Manager.

Clearing All Graphics

This function allows you to delete *all* graphics information for the current circuit, including node, element, and wire coordinates, and the stored schematic. This is different from packing a circuit in that packing a circuit will only delete the stored schematic, *not* the coordinates. (For more information on packing, refer to “Packing the Current Circuit” in Chapter 3.) This function is useful primarily if you need to free memory in your HP 48SX without removing entire circuits. (For more information, refer to “Memory Requirements” in Chapter 1.)

Chapter 8

Data File Manager




The Data File Manager is used to load stored data files, delete unwanted data files, or transfer data files to and from a PC, Macintosh, or other computer.

This chapter covers:

- ☐ Using the Data File Manager
- ☐ Loading a Data File
- ☐ Deleting a Data File
- ☐ Transferring a Data File

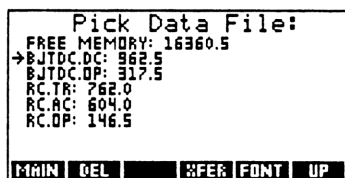
Using the Data File Manager

To get to the Data File Manager, follow these steps:

- ❶ Press   to display all libraries available to your HP 48SX.
- ❷ Find and press **SPICE** to enter the Spice48 Pac library directory.
- ❸ Press the first softkey, **SPICE**, to start the Spice48 Pac.
- ❹ At the Main menu, make sure the pointer is at “Data File Manager” and press .

Items in the Data File Manager

The items listed in the Data File Manager are the data files currently stored in the 'fileS' subdirectory of the 'SPICE48D' directory of your HP 48SX. Data files that were archived to a computer and then deleted will not be shown. The following screen includes tutorial data files (For more information, refer to Chapter 2, “Spice48 Tutorial.”):



After further use of the Spice48 Pac, the Data File Manager will also include your data files, or if you have no data files stored, the screen will display, “Press ADD.”

At the top of the screen is indicated the free memory in your HP 48SX, and next to each data file is indicated the amount of free memory it currently occupies in the 'files' directory.

Summary of Operations

Screen	Softkeys
Data File Manager	MAIN DEL [] XFER FONT UP

Key	Action
DEL	Deletes the data file currently selected by the pointer.
FONT	Toggles between the small and large fonts.
MAIN	Returns to the Main menu.
XFER	Transfers a data file to or from a computer.
UP	Moves up one level in the menu structure, returning to the Main menu.
	Quits the Spice48 Pac to the HP 48SX stack.
	Loads the data file currently selected by the pointer for viewing in the Output Manager.
VIST	Displays the entire text of an item too wide to fit on the screen, up to one entire screen size. If the item fits on the screen, this key is non-functional.
	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Loading a Data File

To load a stored data file for viewing, go to the Data File Manager, move the pointer to the desired data file, and press **ENTER**. This loads in the selected data file and activates the Output Manager. (For more information, refer to Chapter 9, "Output Manager.")

Deleting a Data File

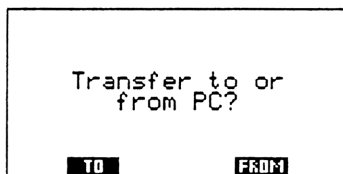
To delete a data file, go to the Data File Manager, move the pointer to the desired data file, and press **DEL**. You will be prompted to verify the delete operation by pressing **YES** to confirm or **NO** or **ATTN** to abort. If you confirm the delete operation, that data file will be deleted.

Transferring a Data File

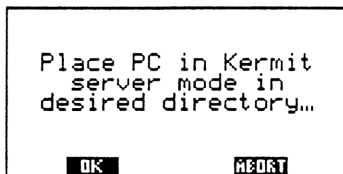
A data file can be transferred either to or from a PC, Macintosh, or other computer with Kermit software and an HP 48SX cable.

Transferring a Data File to a Computer

To download a data file to a computer, go to the Data File Manager, move the pointer to the desired data file, and press **XFER**:



At this prompt, press **TO**:



Now, you must prepare your computer to receive a file with Kermit protocol. For a PC, this involves running MS-KERMIT. For a Macintosh, this involves running a terminal emulator supporting Kermit or running the public domain

version of Kermit. For other computers, this involves running some public domain version of Kermit. In any case, you must set the transfer directory on the computer to the directory to which you want the data file transferred. Then, you must either prepare the computer to receive a single file or place the computer in Kermit server mode. When you have done this, press **OK** to download the data file. To abort the transfer, press **ABORT** or **ATTN**. After the transfer has completed, you may cancel Kermit server mode on your computer and quit the Kermit program.

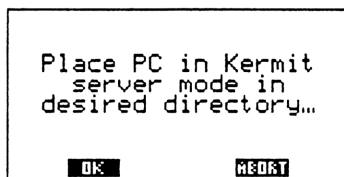
If you successfully download a data file to your computer from your HP 48SX, it will be stored into an ASCII text file in the transfer directory. The data file will remain on your HP 48SX, and if you have transferred the data file for archival purposes, you can now safely delete it by pressing **DEL**. (For more information on deleting a data file, refer above to “Deleting a Data File.”)

Transferring a Data File from a Computer

To upload a data file from a computer, go to the Data File Manager, press **XFER**, and press **FROM**. You will be prompted to enter the name of the data file you wish to download from the computer:



Press **ENTER** when you have finished typing the name:



Now, you must prepare your computer to be a Kermit server. For a PC, this involves running MS-KERMIT. This involves running a public domain version of Kermit. You must set the transfer directory on the computer to the directory in which the circuit you want resides. Then, place the computer in Kermit server mode. When you have done this, press **OK** to upload the data file. To abort the transfer, press **ABORT** or **ATTN**. After the transfer has completed, you may cancel Kermit server mode on your computer and quit the Kermit program.

If you successfully upload a data file to your HP 48SX from your computer, it will be stored in the 'fileS' subdirectory of the 'SPICE48D' directory and listed in the Data File Manager.

Chapter 9

Output Manager


The Output Manager is used to view, plot, and save analysis data. The Output Manager does not appear in the Main menu, but is called when an analysis completes or when you load a stored data file.

This chapter covers:

- ☐ Using the Output Manager
- ☐ Understanding Data Slices
- ☐ Output Manager Views
- ☐ Data View
- ☐ Node View
- ☐ Source Value View
- ☐ Frequency View
- ☐ Time View
- ☐ Node Plots
- ☐ Value Plots
- ☐ Display Forms

Using the Output Manager

To get to the Output Manager, follow these steps:

- ❶ Press  to display all libraries available to your HP 48SX.
- ❷ Find and press **SPICE** to enter the Spice48 Pac library directory.
- ❸ Press the first softkey, **SPICE**, to start the Spice48 Pac.
- ❹ The Output Manager cannot be accessed without data to view. To get data, you must do one of the following:
 - ☐ Perform an analysis of the current circuit. To do this, make sure the pointer is at “Analysis Manager” and press **ENTER**. Then, move the pointer to the desired analysis type and press **ENTER**. Then, specify the analysis parameters and press **CALC** to perform the calculation. The

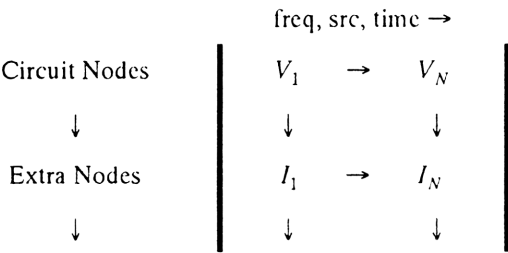
resulting analysis data will be displayed by the Output Manager in an appropriate view. (For more information, refer to Chapter 5, “Analysis Manager.”)

- ❑ Load a stored data file for viewing. To do this, make sure the pointer is at “Data File Manager” and press **ENTER**. Then, move the pointer to the desired data file and press **ENTER**. The selected analysis data will be displayed by the Output Manager in an appropriate view. (For more information, refer to Chapter 8, “Data File Manager.”)

Understanding Data Slices

In general, analysis data created by Spice48 takes the form of a matrix, where the row(s) of the matrix correspond to circuit nodes (for voltages) and extra nodes (for currents) and the column(s) of the matrix correspond to frequencies (for OP or AC analyses), source values (for DC analyses), or times (for TR analyses). An OP analysis always has exactly one column, corresponding to a single frequency, but DC, AC, and TR analyses may have any number of columns, depending on the analysis parameters. (TF analysis data is a special case, and always consists of the same number of values, regardless of circuit size. Therefore, this section is not applicable to TF analysis data.)

The data matrix can be visualized as follows, where N is the number of columns in the matrix (i.e., the number of frequencies, source values, or times):



A *data slice* is any single row or column extracted from this matrix and viewed in the Data View.

The Node View displays the row labels of this matrix, the *output nodes*. The output nodes consist of “V(node)” for each circuit node (representing the voltage at that circuit node) and “I(node)” for each extra node (representing the current at that extra node). At the Node View, you can select one of these output nodes and press **ENTER** to extract the corresponding row from this matrix and view it in the Data View. This is called a *node slice*, and the values displayed in the Data View will consist of the variation of the selected output node as a function of frequency, source value, or time.

The Frequency, Source Value, and Time Views are all similar in that they display the column labels of this matrix, the frequencies, source values, or times, as appropriate. At any of these views, you can select a frequency, source value, or time, and press **ENTER** to extract the corresponding column from this matrix and view it in the Data View. This is called a *frequency slice* or a *source value slice* or a *time slice*, as appropriate, and the values displayed in the Data View will consist of the voltages (for all circuit nodes) and currents (for all extra nodes) for the selected frequency, source value, or time.

Output Manager Views

As indicated above, there are many possible ways to view analysis data in Spice48. The Output Manager always defaults to a view appropriate to the type of analysis data you create or load. In more detail, the different views are:

- ☐ **Data View:** Displays a data slice of node voltages and/or currents. Used for all types of analysis data. Default view for OP and TF analysis data.
- ☐ **Node View:** Displays the output nodes. Used for DC, AC, and TR data. Default view for DC, AC, and TR analysis data. Pressing **ENTER** displays a node slice in the Data View.
- ☐ **Source Value View:** Displays values of the swept source. Used only for DC analysis data. Pressing **ENTER** displays a source value slice in the Data View.
- ☐ **Frequency View:** Displays swept frequencies. Used only for AC analysis data. Pressing **ENTER** displays a frequency slice in the Data View.
- ☐ **Time View:** Displays times at which analysis data was saved. Used only for TR analysis data. Pressing **ENTER** displays a time slice in the Data View.

Data View

The Data View displays a data slice of node voltages and/or currents. It is the only view for OP and TF analysis data, but is one of several possible views for DC, AC, and TR analysis data.

To access the Data View from the Node View, Source Value View, Frequency View, or Time View (for DC, AC, or TR analysis data, as appropriate), move

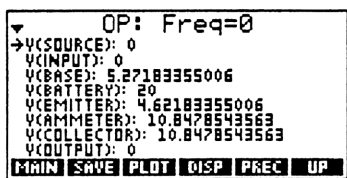
the cursor to the desired output node, source value, frequency, or time and press **ENTER**. To return from the Data View to the previous view, press **UP**. (For more information, refer below to the appropriate section.)

Understanding the Data View

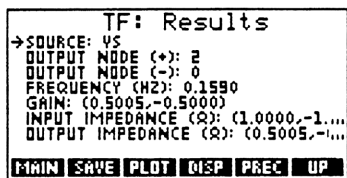
The title and row labels displayed in the Data View depend on the type of analysis data and on which data slice you are viewing:

- ☐ **OP Analysis Data:** Always a frequency slice, so title is “OP: Freq=??”, indicating the frequency corresponding to the displayed data. Row labels are output nodes. The following screen is from tutorial data file 'BJTDC.OP' (For more information, refer to Chapter 2, “Spice48 Tutorial.”):

Frequency Slice



- ☐ **TF Analysis Data:** Slice is not applicable. Row labels are specific to TF analysis data. The following screen is from a TF analysis on tutorial circuit RC (For more information, refer to Chapter 2, “Spice48 Tutorial.”):



- ☐ **DC Analysis Data:** Either a node slice (chosen from Node View), in which case title is “DC: Node ??”, indicating the output node corresponding to the displayed data, or a source value slice (chosen from Source Value View), in which case title is “DC: Source=??”, indicating the source value corresponding to the displayed data. Row labels are either source values for a node slice, or output nodes for a source value slice. The following screens are from tutorial data file 'BJTDC.DC' (For more information, refer to Chapter 2, “Spice48 Tutorial.”):

Source Value Slice

```
▼ DC: VBE=.65
→V(SOURCE): 0
V(INPUT): 0
V(BASE): 5.27183355006
V(BATTERY): 20
V(EMITTER): 4.62183355006
V(AMMETER): 10.8478543563
V(COLLECTOR): 10.8478543563
V(OUTPUT): 0
MAIN SAVE PLOT DISP PREC UP
```

- ❑ **AC Analysis Data:** Either a node slice (chosen from Node View), in which case title is “AC: Node ??”, indicating the output node corresponding to the displayed data, or a frequency slice (chosen from Frequency View), in which case title is “AC: Freq=?”, indicating the frequency corresponding to the displayed data. Row labels are either frequencies for a node slice, or output nodes for a frequency slice. The following screens are from tutorial data file 'RC.AC' (For more information, refer to Chapter 2, “Spice48 Tutorial.”):

Node Slice:

```
▼ AC: Node V(2)
→0.0159: -0.0431
0.0283: -0.1350
0.0503: -0.4132
0.0894: -1.1313
0.1590: -3.0061
0.2827: -6.1869
0.5028: -10.4062
0.8941: -15.1270
MAIN SAVE PLOT DISP PREC UP
```

- ❑ **TR Analysis Data:** Either a node slice (chosen from Node View), in which case title is “TR: Node ??”, indicating the output node corresponding to the displayed data, or a time slice (chosen from Time View), in which case title is “TR: Time=?”, indicating the time corresponding to the displayed data. Row labels are either times for a node slice, or output nodes for a time slice. The following screens are from tutorial data file 'RC.TR' (For more information, refer to Chapter 2, “Spice48 Tutorial.”):

Time Slice:

```
▼ TR: Time=2.0000
→V(1): 0.0000
V(2): 2.1593
I(V5): 2.1593
MAIN SAVE PLOT DISP PREC UP
```

Using the Data View

The Data View provides a convenient way to manipulate analysis data in the form of a single data slice. You can view the data slice in various display forms, plot the data slice, or save the data slice to your HP 48SX for later use. (For information about display forms, refer below to “Display Forms.”)

Plotting the Data Slice

To plot the data slice displayed in the Data View, press **PLOT**.

There are two different types of plots accessible from the Data View—node plots and value plots—but the choice will be made automatically by Spice48, depending on which data slice you are viewing:

- ☐ A node plot will occur if you are viewing a source value slice, a frequency slice, or a time slice.
- ☐ A value plot will occur if you are viewing a node slice.

(For more information on slices, refer above to “Understanding Data Slices,” for more information on node plots, refer below to “Node Plots,” and for more information on value plots, refer below to “Value Plots.”)

Saving the Data Slice

To save the data slice displayed in the Data View, press **SAVE**. You will be prompted to edit the default file name:



```
PRG
{ HOME SPARCOM SPICE48D FILES }
Edit file name:
→ name
DEFAULT.NAME
<SKIP> <SKIP> <DEL> <DEL> <INS> <STK>
```

When you have finished, press **ENTER** to accept the name and save the data slice, or to abort saving the data slice, press **ATN** to clear the command line (if necessary) and then press **ENTER** or **ATN** to return to the Data View without saving the data slice. (For more information on using the editing softkeys, refer to “Text Editing” in Chapter 1.) The data slice will be stored into the file you specify in the ‘fileS’ subdirectory of the ‘SPICE48D’ directory. (For more information, refer to Chapter 8, “Data File Manager.”)

Exiting the Data View




To exit the Data View, press **UP** to return to the previous view or manager, **MAIN** to return to the Main menu, or **ATN** to quit Spice48.

If you accessed the Data View from the Node View, Source Value View, Frequency View, or Time View, pressing **UP** will return to that view.

If you accessed the Data View as a result of loading a data file, pressing **UP** will return to the Data File Manager; if as a result of performing an OP or TF analysis, pressing **UP** will return to the Analysis Manager.

Summary of Operations

Screen	Softkeys					
Data View	MAIN	SAVE	PLOT	DISP	PREC	UP
	MAIN	→STK	PRINT	MEM	FONT	UP

Key	Action
DISP	Prompts for the display type.
FONT	Toggles between the small and large fonts.
MAIN	Returns to the Main menu.
MEM	Displays the size of the displayed data slice and the available free memory.
PLOT	Plots the displayed data slice in a node plot or value plot.
PREC	Prompts for the display precision.
PRINT	Prompts for ONE or ALL to select items, and then sends those items to an IR printer.
SAVE	Saves the displayed data slice.
→STK	Prompts for ONE or ALL to select items, and then copies those items to the stack. The items are placed in a list if ALL was chosen.
UP	Moves up one level in the menu structure, returning to the previous view or manager.
	Quits the Spice48 Pac to the HP 48SX stack.
	Displays the title, the label, and the value, all expanded to one screen.
	Displays the entire text of an item too wide to fit on the screen, up to one entire screen size. If the item fits on the screen, this key is non-functional.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Node View

The Node View displays the output nodes, which consist of “V(node)” for each circuit node (representing the voltage at that circuit node) and “I(node)” for each extra node (representing the current at that extra node). It is the default view for DC, AC, and TR analysis data, and cannot be accessed for OP or TF analysis data. The following screen is from tutorial data file 'BJTDC.DC' (For more information, refer to Chapter 2, “Spice48 Tutorial.”):

DC: Nodes		
V(SOURCE)	V(INPUT)	V(RES)
V(BATTERY)	V(EMITTER)	V(METER)
V(COLLECT)	V(OUTPUT)	V(S)
I(VS)	I(VAMTR)	I(VBATT)
MAIN	MARK	PLOT NODES
SEC	UP	

Using the Node View

Recall that a node slice contains the voltages (for a circuit node) or currents (for an extra node) as a function of frequency, source value, or time, as appropriate to the analysis type. (For more information, refer above to “Understanding Data Slices.”)

The Node View provides a convenient way to manipulate DC, AC, and TR analysis data in the form of node slices. You can view node slices (by switching to the Data View), plot node slices, or save node slices to your HP 48SX for later use. You can mark multiple output nodes to create overlay plots or to save subsets of the analysis data. You can also change display forms. (For information about display forms, refer below to “Display Forms.”)

Marking Nodes

To mark an output node, move the cursor to the desired unmarked output node and press **MARK**. A triangular tag will be added to the output node. Marked output nodes will be used by **PLOT**, **SAVE**, and **MEM** as follows:

- ☐ **PLOT** will overlay node slices for all marked output nodes. (If no output nodes are marked, **PLOT** plots only one node slice.)
- ☐ **SAVE** will extract and save node slices for all marked output nodes. (If no output nodes are marked, **SAVE** saves all node slices.)
- ☐ **MEM** will estimate data file size based on node slices for all marked output nodes. (If no output nodes are marked, **MEM** bases estimate on all node slices.)

The **MARK** softkey is actually a toggle, so to unmark an output node, move the cursor to the desired marked output node and press **MARK**. The triangular tag will be removed from the output node.

Viewing a Node Slice

To view a node slice, move the cursor to the desired output node and press **ENTER**. This will display the node slice in the Data View. When you have finished viewing the node slice, press **UP** to return to the Node View. Only one node slice can be viewed at a time, so marked output nodes are ignored—the output node currently selected by the cursor is the one that is viewed. (For more information, refer above to “Data View.”)

Note: Node slices displayed in the Data View will include only marked frequencies, source values, or times, unless none are marked, in which case all will be included. (For more information, refer below to “Frequency View,” “Source Value View,” or “Time View.”)

Plotting Node Slice(s)

To plot multiple node slices, first mark the desired output nodes and then press **PLOT**. This will create a value plot of the marked node slices. To plot a single node slice, make sure either no output nodes are marked or only the desired output node is marked and press **PLOT**. This will create a value plot of the desired node slice. (For more information, refer below to “Value Plots.”)

Note: Plotted node slices will include only marked frequencies, source values, or times, unless none are marked, in which case all will be included. (For more information, refer below to “Frequency View,” “Source Value View,” or “Time View.”)

Saving Node Slice(s)

To save all node slices, make sure either no output nodes are marked or all output nodes are marked, or to save only some of the node slices, mark the desired output nodes. When you have finished, press **SAVE**. You will be prompted to edit the default file name. When you have finished, press **ENTER** to accept the name and save the node slice(s), or to abort saving the node slice(s), press **ATTN** to clear the command line (if necessary) and then press **ENTER** or **ATTN** to return to the Node View without saving the node slice(s). (For more information on using the editing softkeys, refer to “Text Editing” in Chapter 1.) The node slice(s) will be stored in the file you specify in the 'fileS' subdirectory of the 'SPICE48D' directory. (For more information, refer to Chapter 8, “Data File Manager.”)

Note: Saved node slices will include only marked frequencies, source values, or times, unless none are marked, in which case all will be included. (For more information, refer below to “Frequency View,” “Source Value View,” or “Time View.”)

Exiting the Node View

To exit the Node View, press **UP** to return to the previous manager, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.

If you accessed the Node View as a result of loading a data file, pressing **UP** will return to the Data File Manager, or if you accessed the Node View as a result of performing a DC, AC, or TR analysis, pressing **UP** will return to the Analysis Manager.

Summary of Operations

Screen	Softkeys					
Node View	MAIN	MARK	PLOT	NODE	????	UP
	MAIN	SAVE	MEM	DISP	PREC	UP

Key	Action
????	Softkey is either SRC , FREQ , or TIME , and switches to either the Source Value View, the Frequency View, or the Time View, as appropriate.
DISP	Prompts for the display type.
MAIN	Returns to the Main menu.
MARK	Toggles output node currently selected by the cursor between marked and unmarked states.
MEM	Displays the size of the marked node slice(s) and the available free memory.
NODE	Does nothing because the Node View is currently displayed.
PLOT	Plots one or more node slices in a value plot.
PREC	Prompts for the display precision.
SAVE	Saves one or more node slices.
UP	Moves up one level in the menu structure, returning to the previous manager.
ATTN	Quits the Spice48 Pac to the HP 48SX stack.
ENTER	Switches to the Data View and displays the node slice of the output node currently selected by the cursor.
ON PREV	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Source Value View

The Source Value View displays the values of the swept source. It can be accessed only for DC analysis data. The following screen is from tutorial data file 'BJTDC.DC' (For more information, refer to Chapter 2, "Spice48 Tutorial."):

DC: VBE Values		
6	.625	.65
.675	7	.725
.75		
MAIN MARK PLOT NOOE SRC= UP		

Using the Source Value View

Recall that a source value slice contains the voltages (for all circuit nodes) and currents (for all extra nodes) for the selected source value. (For more information, refer above to "Understanding Data Slices.")

The Source Value View provides a convenient way to manipulate DC analysis data in the form of source value slices. You can view source value slices (by switching to the Data View), plot source value slices, or save source value slices to your HP 48SX for later use. You can mark multiple source values to create overlay plots or to save subsets of the analysis data. You can also change display forms. (For information about display forms, refer below to "Display Forms.")

Marking Source Values

To mark a source value, move the cursor to the desired unmarked source value and press **MARK**. A triangular tag will be added to the source value. Marked source values will be used by **PLOT**, **SAVE**, and **MEM** as follows:

- ☐ **PLOT** will overlay source value slices for all marked source values. (If no source values are marked, **PLOT** plots only one source value slice.)
- ☐ **SAVE** will extract and save source value slices for all marked source values. (If no source values are marked, **SAVE** saves all source value slices.)
- ☐ **MEM** will estimate data file size based on source value slices for all marked source values. (If no source values are marked, **MEM** bases estimate on all source value slices.)

The **MARK** softkey is actually a toggle, so to unmark a source value, move the cursor to the desired marked source value and press **MARK**. The triangular tag will be removed from the source value.

Viewing a Source Value Slice

To view a source value slice, move the cursor to the desired source value and press **ENTER**. This will display the source value slice in the Data View. When you have finished viewing the source value slice, press **UP** to return to the Source Value View. Only one source value slice can be viewed at a time, so marked source values are ignored—the source value currently selected by the cursor is the one that is viewed. (For more information, refer above to “Data View.”)

Note: Source value slices displayed in the Data View will include only marked output nodes, unless no output nodes are marked, in which case all output nodes will be included. (For more information, refer above to “Node View.”)

Plotting Source Value Slice(s)

To plot multiple source value slices, first mark the desired source values and then press **PLOT**. This will create a node plot of the marked source value slices. To plot a single source value slice, make sure either no source values are marked or only the desired source value is marked and press **PLOT**. This will create a node plot of the desired source value slice. (For more information, refer below to “Node Plots.”)

Note: Plotted source value slices will include only marked output nodes, unless no output nodes are marked, in which case all output nodes will be included. (For more information, refer above to “Node View.”)

Saving Source Value Slice(s)

To save all source value slices, make sure either no source values are marked or all source values are marked, or to save only some of the source value slices, mark the desired source values. When you have finished, press **SAVE**. You will be prompted to edit the default file name. When you have finished, press **ENTER** to accept the name and save the source value slice(s), or to abort saving the source value slice(s), press **ATTN** to clear the command line (if necessary) and then press **ENTER** or **ATTN** to return to the Source Value View without saving the source value slice(s). (For more information on using the editing softkeys, refer to “Text Editing” in Chapter 1.) The source value slice(s) will be stored into the file you specify in the 'fileS' subdirectory of the 'SPICE48D' directory. (For more information, refer to Chapter 8, “Data File Manager.”)

Note: Saved source value slices will include only marked output nodes, unless no output nodes are marked, in which case all output nodes will be included. (For more information, refer above to “Node View.”)

Exiting the Source Value View

To exit the Source Value View, press **UP** to return to the previous manager, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.

If you accessed the Source Value View as a result of loading a data file, pressing **UP** will return to the Data File Manager, or if you accessed the Source Value View as a result of performing a DC analysis, pressing **UP** will return to the Analysis Manager.

Summary of Operations

Screen	Softkeys					
Source Value View	MAIN	MARK	PLOT	NODE	SRC	UP
	MAIN	SAVE	MEM	DISP	PREC	UP

Key	Action
DISP	Prompts for the display type.
MAIN	Returns to the Main menu.
MARK	Toggles source value currently selected by the cursor between marked and unmarked states.
MEM	Displays the size of the marked source value slice(s) and the available free memory.
NODE	Switches to the Node View.
PLOT	Plots one or more source value slices in a node plot.
PREC	Prompts for the display precision.
SAVE	Saves one or more source value slices.
SRC	Does nothing because the Source Value View is currently displayed.
UP	Moves up one level in the menu structure, returning to the previous manager.
ATTN	Quits the Spice48 Pac to the HP 48SX stack.
ENTER	Switches to the Data View and displays the source value slice of the source value currently selected by the cursor.
ON — PREV	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Frequency View

The Frequency View displays the swept frequencies. It can be accessed only for AC analysis data. The following screen is from tutorial data file 'RC.AC' (For more information, refer to Chapter 2, "Spice48 Tutorial."):

AC: Frequencies		
0.159	2.8227464261	5.028021479
8.941227070	1.59	2.8227464261
5.028021479	8.941227070	1.59
MAIN MARK PLOT MODE FREQ UP		

Using the Frequency View

Recall that a frequency slice contains the voltages (for all circuit nodes) and currents (for all extra nodes) for the selected frequency. (For more information, refer above to "Understanding Data Slices.")

The Frequency View provides a convenient way to manipulate AC analysis data in the form of frequency slices. You can view frequency slices (by switching to the Data View), plot frequency slices, or save frequency slices to your HP 48SX for later use. You can mark multiple frequencies to create overlay plots or to save subsets of the analysis data. You can also change display forms. (For information about display forms, refer below to "Display Forms.")

Marking Frequencies

To mark a frequency, move the cursor to the desired unmarked frequency and press **MARK**. A triangular tag will be added to the frequency. Marked frequencies will be used by **PLOT**, **SAVE**, and **MEM** as follows:

- ☐ **PLOT** will overlay frequency slices for all marked frequencies. (If no frequencies are marked, **PLOT** plots only one frequency slice.)
- ☐ **SAVE** will extract and save frequency slices for all marked frequencies. (If no frequencies are marked, **SAVE** saves all frequency slices.)
- ☐ **MEM** will estimate data file size based on frequency slices for all marked frequencies. (If no frequencies are marked, **MEM** bases estimate on all frequency slices.)

The **MARK** softkey is actually a toggle, so to unmark a frequency, move the cursor to the desired marked frequency and press **MARK**. The triangular tag will be removed from the frequency.

Viewing a Frequency Slice

To view a frequency slice, move the cursor to the desired frequency and press **ENTER**. This will display the frequency slice in the Data View. When you have finished viewing the frequency slice, press **UP** to return to the Frequency View. Only one frequency slice can be viewed at a time, so marked frequencies are ignored—the frequency currently selected by the cursor is the one that is viewed. (For more information, refer above to “Data View.”)

Note: Frequency slices displayed in the Data View will include only marked output nodes, unless no output nodes are marked, in which case all output nodes will be included. (For more information, refer above to “Node View.”)

Plotting Frequency Slice(s)

To plot multiple frequency slices, first mark the desired frequencies and then press **PLOT**. This will create a node plot of the marked frequency slices. To plot a single frequency slice, make sure either no frequencies are marked or only the desired frequency is marked and press **PLOT**. This will create a node plot of the desired frequency slice. (For more information, refer below to “Node Plots.”)

Note: Plotted frequency slices will include only marked output nodes, unless no output nodes are marked, in which case all output nodes will be included. (For more information, refer above to “Node View.”)

Saving Frequency Slice(s)

To save all frequency slices, make sure either no frequencies are marked or all frequencies are marked, or to save only some of the frequency slices, mark the desired frequencies. When you have finished, press **SAVE**. You will be prompted to edit the default file name. When you have finished, press **ENTER** to accept the name and save the frequency slice(s), or to abort saving the frequency slice(s), press **ATTN** to clear the command line (if necessary) and then press **ENTER** or **ATTN** to return to the Frequency View without saving the frequency slice(s). (For more information on using the editing softkeys, refer to “Text Editing” in Chapter 1.) The frequency slice(s) will be stored into the file you specify in the 'fileS' subdirectory of the 'SPICE48D' directory. (For more information, refer to Chapter 8, “Data File Manager.”)

Note: Saved frequency slices will include only marked output nodes, unless no output nodes are marked, in which case all output nodes will be included. (For more information, refer above to “Node View.”)

Exiting the Frequency View

To exit the Frequency View, press **UP** to return to the previous manager, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.

If you accessed the Frequency View as a result of loading a data file, pressing **UP** will return to the Data File Manager, or if you accessed the Frequency View as a result of performing an AC analysis, pressing **UP** will return to the Analysis Manager.

Summary of Operations

Screen	Softkeys						
Frequency View	MAIN	MARK	PLOT	NODE	FREQ	UP	
	MAIN	SAVE	MEM	DISP	PREC	UP	

Key	Action
DISP	Prompts for the display type.
FREQ	Does nothing because the Frequency View is currently displayed.
MAIN	Returns to the Main menu.
MARK	Toggles frequency currently selected by the cursor between marked and unmarked states.
MEM	Displays the size of the marked frequency slice(s) and the available free memory.
NODE	Switches to the Node View.
PLOT	Plots one or more frequency slices in a node plot.
PREC	Prompts for the display precision.
SAVE	Saves one or more frequency slices.
UP	Moves up one level in the menu structure, returning to the previous manager.
ATTN	Quits the Spice48 Pac to the HP 48SX stack.
ENTER	Switches to the Data View and displays the frequency slice of the frequency currently selected by the cursor.
ON — PREV	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Time View

The Time View displays times at which analysis data was saved. It can be accessed only for TR analysis data. The following screen is from tutorial data file 'RC.TR' (For more information, refer to Chapter 2, "Spice48 Tutorial."):

TR: Times		
0	2	4
6	8	1
1.2	1.4	1.6
1.8	2	2.2
2.4	2.6	2.8
MAIN	MARK	PLOT
NODE	TIME	UP

Using the Time View

Recall that a time slice contains the voltages (for all circuit nodes) and currents (for all extra nodes) for the selected time. (For more information, refer above to "Understanding Data Slices.")

The Time View provides a convenient way to manipulate TR analysis data in the form of time slices. You can view time slices (by switching to the Data View), plot time slices, or save time slices to your HP 48SX for later use. You can mark multiple times to create overlay plots or to save subsets of the analysis data. You can also change display forms. (For information about display forms, refer below to "Display Forms.")

Marking Times

To mark a time, move the cursor to the desired unmarked time and press **MARK**. A triangular tag will be added to the time. Marked times will be used by **PLOT**, **SAVE**, and **MEM** as follows:

- ☐ **PLOT** will overlay time slices for all marked times. (If no times are marked, **PLOT** plots only one time slice.)
- ☐ **SAVE** will extract and save time slices for all marked times. (If no times are marked, **SAVE** saves all time slices.)
- ☐ **MEM** will estimate data file size based on time slices for all marked times. (If no times are marked, **MEM** bases estimate on all time slices.)

The **MARK** softkey is actually a toggle, so to unmark a time, move the cursor to the desired marked time and press **MARK**. The triangular tag will be removed from the time.

Viewing a Time Slice

To view a time slice, move the cursor to the desired time and press **ENTER**. This will display the time slice in the Data View. When you have finished viewing the time slice, press **UP** to return to the Time View. Only one time slice can be viewed at a time, so marked times are ignored—the time currently selected by the cursor is the one that is viewed. (For more information, refer above to “Data View.”)

Note: Time slices displayed in the Data View will include only marked output nodes, unless no output nodes are marked, in which case all output nodes will be included. (For more information, refer above to “Node View.”)

Plotting Time Slice(s)

To plot multiple time slices, first mark the desired times and then press **PLOT**. This will create a node plot of the marked time slices. To plot a single time slice, make sure either no times are marked or only the desired time is marked and press **PLOT**. This will create a node plot of the desired time slice. (For more information, refer below to “Node Plots.”)

Note: Plotted time slices will include only marked output nodes, unless no output nodes are marked, in which case all output nodes will be included. (For more information, refer above to “Node View.”)

Saving Time Slice(s)

To save all time slices, make sure either no times are marked or all times are marked, or to save only some of the time slices, mark the desired times. When you have finished, press **SAVE**. You will be prompted to edit the default file name. When you have finished, press **ENTER** to accept the name and save the time slice(s), or to abort saving the time slice(s), press **ATTN** to clear the command line (if necessary) and then press **ENTER** or **ATTN** to return to the Time View without saving the time slice(s). (For more information on using the editing softkeys, refer to “Text Editing” in Chapter 1.) The time slice(s) will be stored into the file you specify in the 'fileS' subdirectory of the 'SPICE48D' directory. (For more information, refer to Chapter 8, “Data File Manager.”)

Note: Saved time slices will include only marked output nodes, unless no output nodes are marked, in which case all output nodes will be included. (For more information, refer above to “Node View.”)

Exiting the Time View

To exit the Time View, press **UP** to return to the previous manager, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.

If you accessed the Time View as a result of loading a data file, pressing **UP** will return to the Data File Manager, or if you accessed the Time View as a result of performing a TR analysis, pressing **UP** will return to the Analysis Manager.

Summary of Operations

Screen	Softkeys					
Time View	MAIN	MARK	PLOT	NODE	TIME	UP
	MAIN	SAVE	MEM	DISP	PREC	UP

Key	Action
DISP	Prompts for the display type.
MAIN	Returns to the Main menu.
MARK	Toggles time currently selected by the cursor between marked and unmarked states.
MEM	Displays the size of the marked time slice(s) and the available free memory.
NODE	Switches to the Node View.
PLOT	Plots one or more time slices in a node plot.
PREC	Prompts for the display precision.
SAVE	Saves one or more time slices.
TIME	Does nothing because the Time View is currently displayed.
UP	Moves up one level in the menu structure, returning to the previous manager.
ATA	Quits the Spice48 Pac to the HP 48SX stack.
ENTER	Switches to the Data View and displays the time slice of the time currently selected by the cursor.
ON PREV	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Node Plots

A *node plot* graphically demonstrates the “state” of the circuit by plotting the values of each output node for one or more selected frequencies (AC analysis data), source values (DC analysis data), or times (TR analysis data). Respectively, these correspond to frequency slices, source value slices, or time slices. (For more information on data slices, refer above to “Understanding Data Slices.”) If more than one data slice is plotted on a node plot, it is an *overlay* node plot.

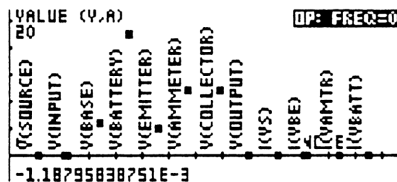
Plotting Ranges

A node plot always includes all output nodes. Therefore, you do not have any control over the horizontal axis data, because it consists of the output nodes, displayed as labels next to the appropriate voltage or current value. However, you will be prompted to set the vertical range. A default vertical range will be suggested which will guarantee that all data appears on the screen. To accept the default vertical range, press **ENTER**, or to adjust the vertical range, use the editing softkeys or press **ATTN** to clear the command line and enter a new vertical range. After editing the vertical range, press **ENTER** to accept the new vertical range and create a node plot, or to abort the node plot, press **ATTN** to clear the command line (if necessary) and then press **ENTER** or **ATTN** to return to the previous view without creating a node plot. (For more information on using the editing softkeys, refer to “Text Editing” in Chapter 1.)

Node Plot

If more than one data slice is plotted, each is assigned a distinct symbol, and a symbol key is drawn to the right of the plot. Node plots with more than 10 output nodes or with a symbol key will be wider than your HP 48SX screen. If this is the case, scrolling will automatically be activated.










The following node plot is from tutorial data file 'BJTDC.OP' (For more information, refer to Chapter 2, “Spice48 Tutorial.”):



When you have finished viewing the node plot, press **ATTN** to return to the previous view.

Summary of Operations

Screen	Softkeys
Node Plot	None

Key	Action
  (Scrolling)	Scrolls node plots wider than the screen to view the extra nodes and/or the symbol key. When prefixed with  , jumps to the edge of the node plot in the indicated direction.
	Exits the node plot and returns to the previous view.
 	Toggles between scrolling mode and Graphics Environment. (Graphics Environment is not useful for node plots.)
	If Graphics Environment is active, copies value plot (PICT) to the HP 48SX stack as a graphics object (GROB).
 	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Value Plots

A *value plot* graphically demonstrates the variation of one or more output node voltages or currents as a function of frequency (AC analysis data), source value (DC analysis data), or time (TR analysis data). If more than one output node is plotted on a value plot, it is an *overlay* value plot.

After you press **PLOT**, you will be prompted to select the type of value plot:

Pick plot type:

POINT(S)
LINE(S)
EQUATION(S)

MAIN
UP

Move the cursor to the desired type and press **ENTER**. (Instead of pressing **ENTER**, you can press **UP** to abort the operation, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.)

Here are specific descriptions of the various types:

- ☐ **Point:** Plots points for each selected node slice. If more than one node slice is plotted, each is assigned a distinct symbol, and a symbol key is drawn to the right of the plot.
- ☐ **Line:** Identical to point, except lines connect the points for each selected node slice. If more than one node slice is plotted, each is assigned a distinct symbol, and a symbol key is drawn to the right of the plot.
- ☐ **Equation:** Creates a piecewise linear representation of the node slice data and uses the built-in HP 48SX function plotting routines to plot the resulting algebraic equation for each node slice. The HP 48SX Graphics Environment function analysis menu can be used for further analysis. Node slices do not receive distinct symbols in equation plots.

(For more information, refer below to the appropriate section.)

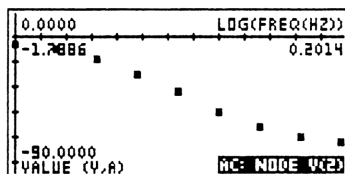
Plotting Ranges

For a value plot, you will be prompted to set both the horizontal and vertical ranges. Default ranges will be suggested which will guarantee that all of the data appears on the screen. To accept a default ranges, press **ENTER**, or to adjust the range, use the editing softkeys or press **ATTN** to clear the command line and enter a new range. After editing the range, press **ENTER** to accept the new range, or to abort the value plot, press **ATTN** to clear the command line (if necessary) and then press **ENTER** or **ATTN** to return to the previous view without creating a value plot. (For more information on using the editing softkeys, refer to “Text Editing” in Chapter 1.)

Point Plot

A point plot displays the variation of one or more output node voltages or currents as a series of points (symbols). If more than one node slice is plotted (an overlay value plot), the data from each node slice is plotted with a distinct symbol, and a symbol key is drawn to the right of the value plot. Point plots with a symbol key will be wider than your HP 48SX screen, so scrolling will automatically be activated.

The following point plot is from tutorial data file 'RC.AC' (For more information, refer to Chapter 2, “Spice48 Tutorial.”):

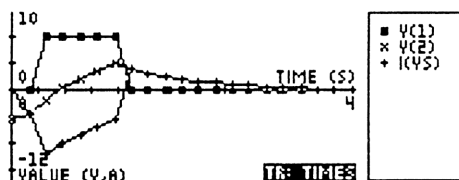


When you have finished viewing the point plot, press **ATTN** to return to the previous view.

Line Plot

A line plot displays the variation of one or more output node voltages or currents as a series of lines connecting points (symbols). If more than one node slice is plotted (an overlay value plot), the data from each node slice is plotted with a distinct symbol, and a symbol key is drawn to the right of the value plot. Line plots with a symbol key will be wider than your HP 48SX screen, so scrolling will automatically be activated.

The following line plot is from tutorial data file 'RC.TR' (For more information, refer to Chapter 2, "Spice48 Tutorial."):



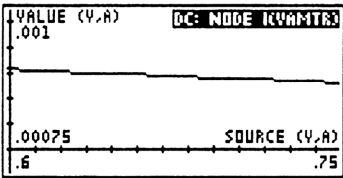
When you have finished viewing the line plot, press **ATTN** to return to the previous view.

Equation Plot

An equation plot displays the variation of one or more output node voltages or currents as a series of piecewise linear functions. These functions are created from the node slice data and plotted by the built-in plotting functions. Because of this, you can use the Graphics Environment to further analyze equation plots. (For more information, refer to Chapter 18 of the HP 48SX Owner's Manual, "Basic Plotting and Function Analysis.")

Because the piecewise linear functions may be large and evaluate slowly, you can increase the plotting resolution. In general, if you expect the data to be linear, a larger plot resolution can be tolerated without adversely affecting the display, whereas if you expect the data to be non-linear, a smaller plot resolution should be used to guarantee an accurate graph.

The following equation plot is from tutorial data file 'BJTDC.DC' (For more information, refer to Chapter 2, "Spice48 Tutorial."):

















Press **GRAPH** to toggle between the default view (scrolling mode) and the Graphics Environment. Useful functions in the Graphics Environment include **SLOPE**, **COORD**, **F'**, and **F·X**. (For more information, refer to Chapter 18 of the HP 48SX Owner's Manual, "Basic Plotting and Function Analysis.")

When you have finished viewing the equation plot, press **ATTN** to return to the previous view.

Summary of Operations

Screen	Softkeys					
Value Plot	None					
Graphics Environment	ZOOM	Z-BOX	CENT	COORD	LABEL	FCN
	DOT+	DOT-	LINE	TLINE	BOX	FIRCL
	MARK	REPL	SUB	DEL	+ -	KEYS
Function Analysis	ROOT	I SECT	SLOPE	AREA	EXTR	EXIT
	F·X	F'	NXEQ			

Key	Action
COORD	Displays the coordinates of the graphics cursor position.
F·X	Displays the function value at the horizontal axis value of the graphics cursor, and moves the graphics cursor to that point on the function. (Equation plots only.)
F'	Plots the first derivative the current function. (Equation plots only.)
FCN	Displays the Function Analysis menu for further analysis. (Function Analysis menu is useful only for equation plots.)
NXEQ	Switches to the next function in the value plot. (Equation plots only.)

SLOPE	Displays the slope of the current function at the horizontal axis value of the graphics cursor, and moves the graphics cursor to the point at which the slope was calculated. (Equation plots only.)
SUB	Copies the rectangle bounded by the mark and the graphics cursor location to the HP 48SX stack as a graphics object (GROB).
    (Graphics)	Moves the graphics cursor in the indicated direction. When prefixed with  , moves the graphics cursor to the edge of the screen in the indicated direction.
  (Scrolling)	Scrolls value plots wider than the screen to view the symbol key. When prefixed with  , jumps to the far edge of the value plot in that direction.
	Exits the value plot and returns to the previous view.
 	Toggles between scrolling mode and Graphics Environment.
	Copies value plot (PICT) to the HP 48SX stack as a graphics object (GROB).
 	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Display Forms

Analysis data can be displayed in many different forms. From any Output Manager view, you can change the display type and/or precision.

Selecting Display Type

To change the display type, press **DISP** at any Output Manager view. You will be prompted for the display type:

Pick display type:			
STANDARD	MAGNITUDE	REAL	
IMAGINARY	PHASE <	DECIBELS	
MIN			UP

Move the cursor to the desired display type and press **ENTER**. (Instead of pressing **ENTER**, you can press **UP** to abort the operation, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.)

Here are specific descriptions of the display types:

- ☐ **Standard:** Displays the standard form of the data, which for complex numbers, will depend on the XYZ / R∠Z (POLAR) setting of your HP 48SX. (For more information, refer to Chapter 11 of the HP 48SX Owner's Manual, "Complex Numbers.")
- ☐ **Magnitude:** Displays the magnitude of the data, or $abs(x)$.
- ☐ **Real:** Displays the real part of the data, or $Re(x)$.
- ☐ **Imaginary:** Displays the imaginary part of the data, or $Im(x)$, which will depend on the XYZ / R∠Z (POLAR) setting of your HP 48SX. (For more info, refer to Chapter 11 of the HP 48SX Owner's Manual.)
- ☐ **Phase ∠:** Displays the polar angle of the data, or $arg(x)$, which will depend on the DEG / RAD / GRAD setting of your HP 48SX. (For more info, refer to Chapter 9 of the HP 48SX Owner's Manual.)
- ☐ **Decibels:** Displays the data in dB, or $20\log_{10}(x)$.

Selecting Display Precision

To change the display precision, press **PREC** at any Output Manager view:

Pick precision:			
STANDARD	FIXED	SCIENTIFIC	
ENGINEER			
MAIN			UP

Move the cursor to the desired display precision type and press **ENTER**. (Instead of pressing **ENTER**, you can press **UP** to abort the operation, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.)

If you select standard, you will not have to specify the number of decimal places. If you select any other display precision type, you will be prompted to enter or edit the number of decimal places. When you have finished, press **ENTER** to accept the new number of decimal places and return to the Output Manager, or to abort changing the display precision, press **ATTN** to clear the command line (if necessary) and then press **ENTER** or **ATTN** to return to the Output Manager.

Chapter 10

Miscellaneous Info


Miscellaneous Info includes a number of useful utility functions, reference data tables, a library of constants, and an ASCII table.

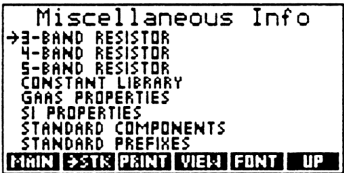
This chapter covers:

- ☐ Using Miscellaneous Info
- ☐ Resistor Colors
- ☐ Constant Library
- ☐ GaAs and Si Properties
- ☐ Standard Components
- ☐ Standard Prefixes

Using Miscellaneous Info

To get to Miscellaneous Info, follow these steps:

- 1 Press  **LIBRARY** to display all libraries available to your HP 48SX.
- 2 Find and press **SPICE** to enter the Spice48 Pac library directory.
- 3 Press the first softkey, **SPICE**, to start the Spice48 Pac.
- 4 At the Main menu, make sure the pointer is at “Miscellaneous Info” and press **ENTER**:



Miscellaneous Info lists various utility functions. A function is selected by moving the pointer to the desired item and pressing **ENTER**.

Items in Miscellaneous Info





Each item in Miscellaneous Info is briefly described below and is discussed in detail in the various sections of this chapter.

Item	Description
3-Band Resistor	Converts 3 color bands to a value and tolerance.
4-Band Resistor	Converts 4 color bands to a value and tolerance.
5-Band Resistor	Converts 5 color bands to a value and tolerance.
Constant Library	Displays 45 commonly used constants.
GaAs Properties	Displays 28 commonly used properties of GaAs.
Si Properties	Displays 28 commonly used properties of Si.
Standard Components	Calculates closest standard component (resistor, inductor, or capacitor) value to desired value.
Standard Prefixes	Displays common prefixes.
ASCII Table	Displays an interactive 256 character ASCII table.

Summary of Operations

Screen	Softkeys
Miscellaneous Info	MAIN ←STK PRINT VIEW FONT UP

Key	Action
FONT	Toggles between the small and large fonts.
MAIN	Returns to the Main menu.
PRINT	Prompts for ONE or ALL to select items, and then sends those items to an IR printer.
←STK	Prompts for ONE or ALL to select items, and then copies those items to the stack. The items are placed in a list if ALL was chosen.
UP	Moves up one level in the menu structure, returning to the Main menu.
VIEW	Displays the entire text of an item too wide to fit on the screen, up to one entire screen size. If the item fits on the screen, this key is non-functional.

	Quits the Spice48 Pac to the HP 48SX stack.
	Moves down one level in the menu structure, entering the selected function.
 	Dumps the current screen to an IR printer.








For a complete summary of operations, refer to Appendix B, "Summary of Operations."




Resistor Colors

Three functions are provided to convert the colors on 3-band, 4-band, and 5-band resistors to the resistor value in ohms and the resistor tolerance. (Inductors and capacitors have no color bands.)



Upon selecting one of these functions, a screen similar to the following appears (the 4-band resistor screen is shown):

[]					
BLACK		BROWN		RED	
ORANGE		YELLOW		GREEN	
BLUE		VIOLET		GREY	
WHITE		SILVER		GOLD	
MAIN		1	2	3	CALC UP

You should now specify colors for the various bands of the resistor. To do this, move the cursor to the desired color and press , , , or  , according to which band is of the selected color. When you have specified all of the bands, press  or  to calculate the resistor value and tolerance. The results will be displayed on a separate screen.

At any point, press  to return to Miscellaneous Info,  to return to the Main menu, or  to quit Spice48.

Resistor Band Descriptions and Restrictions

You can display a reference chart indicating the color restrictions for each band by pressing  . There are three types of resistor bands, each of which has various restrictions:

- ☐ **Digit Bands:** 1st and 2nd bands for 3- and 4-band resistors; 1st, 2nd, and 3rd bands for 5-band resistors. Digit bands cannot be silver or gold.

- ☐ **Exponent Band:** 3rd band for 3- and 4-band resistors; 4th band for 5-band resistors. The exponent band can be of any color.
- ☐ **Tolerance Band:** 4th band for 4-band resistors; 5th band for 5-band resistors. (3-band resistors have no tolerance band; their tolerance is assumed to be $\pm 20\%$.) The tolerance band cannot be black, orange, yellow, or white.

If a color band on the resistor you are converting conflicts with any of the above rules, you may have the resistor reversed—try entering the colors from the opposite direction.

Summary of Operations

Screen	Softkeys					
Resistor Colors	MAIN	1	2	3	CALC	UP
	MAIN	4	5	PICT	CALC	UP

Key	Action
1	Sets 1st band to the color currently selected by the cursor.
2	Sets 2nd band to the color currently selected by the cursor.
3	Sets 3rd band to the color currently selected by the cursor.
4	Sets 4th band to the color currently selected by the cursor.
5	Sets 5th band to the color currently selected by the cursor.
CALC	Calculates the resistor value and tolerance from the colors.
FONT	Toggles between the small and large fonts.
PICT	Displays a reference chart indicating the color restrictions.
PRINT	Prompts for ONE or ALL to select items, and then sends those items to an IR printer.
-STK	Prompts for ONE or ALL to select items, and then copies those items to the stack. The items are placed in a list if ALL was chosen.
UP	Moves up one level in the menu structure, returning to Miscellaneous Info.
VIEW	Displays the entire text of an item too wide to fit on the screen, up to one entire screen size. If the item fits on the screen, this key is non-functional.
ATTN	Quits the Spice48 Pac to the HP 48SX stack.

ENTER	At the resistor color entry screen, calculates the resistor value and tolerance from the colors. At the resistor color results screen, displays the screen title, the item label, and the item, all expanded to one screen.
ON PREV	Dumps the current screen to an IR printer.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

Constant Library

Upon selecting this function, the constant library appears:

Constant Library	
→PI	CIRCLE RATIO : 3.1415926...
E	NAPIERE CONSTANT : 2.71828...
γ	EULER CONSTANT : 0.57721...
φ	GOLDEN RATIO : 1.61803...
α	FINE STRUCTURE : .007297...
C	SPEED OF LIGHT : 2.997924...
ε0	PERMITTIVITY : 8.8541878...
F	FARADAY CONSTANT : 9.64845...
MAIN ←STK PRINT UNITS FONT UP	






Browse through the list to find the desired constant, or use the search mode. (For more information on using the search mode, refer to “Using the Search Mode” in Chapter 1.) When you have found the desired constant, press **ENTER** to display the constant description and value on a full screen, press **→STK** to copy the value to the stack, or press **PRINT** to print the value on an IR printer.

When you have finished browsing the Constant Library, press **UP** to return to Miscellaneous Info, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.

Summary of Operations

Screen	Softkeys
Constant Library	MAIN →STK PRINT UNITS FONT UP


Key	Action
FONT	Toggles between the small and large fonts.
MAIN	Returns to the Main menu.
PRINT	Prompts for ONE or ALL to select items, and then sends those items to an IR printer.


→STK	Prompts for ONE or ALL to select items, and then copies those items to the stack. The items are placed in a list if ALL was chosen.
UNITS	Pressing this key toggles units, stripping or appending units to all values.
UP	Moves up one level in the menu structure, returning to the Main menu.
	Quits the Spice48 Pac to the HP 48SX stack.
	Displays the screen title, the constant label, and the constant value, all expanded to one screen.
 	Dumps the current screen to an IR printer.
	Displays the entire text of an item too wide to fit on the screen, up to one entire screen size. If the item fits on the screen, this key is non-functional.

For a complete summary of operations, refer to Appendix B, “Summary of Operations.”

GaAs and Si Properties

Upon selecting one of these functions, a screen containing 28 commonly used properties of GaAs or Si will appear.

Browse through the list to find the desired property, or use the search mode. (For more information on using the search mode, refer to “Using the Search Mode” in Chapter 1.) When you have found the desired property, press  to display the property description and value on a full screen, press **→STK** to copy the value to the stack, or press **PRINT** to print the value on an IR printer.

When you have finished browsing the property list, press **UP** to return to Miscellaneous Info, **MAIN** to return to the Main menu, or  to quit Spice48.

Standard Components

This function calculates the closest available standard component value to the value you enter at the tolerance you specify. Components supported include resistors, inductors, and capacitors, and the color bands are provided (if possible) in case the desired component is a resistor.

Upon selecting this function, you will be prompted for the desired component value:

```
PRG
{ HOME SPARCOM SPICE48D }
Enter component value:
→ real(1E-15-1E12)
↓
TERM GIGA MEGA KILO MILLI MICRO
```

You should now enter the desired component value, using the softkeys to append an exponent, and press **ENTER** to accept the value. (To abort the operation, press **ATTN** to clear the command line (if necessary) and then press **ENTER** or **ATTN** to return to Miscellaneous Info without calculating a standard component value.)

After you enter the desired component value and press **ENTER**, you will be prompted for the desired tolerance:

Pick tolerance:		
±20%	±2%	±0.25%
±10%	±1%	±0.10%
±5%	±0.5%	±0.05%
MAIN		UP

You should now select the tolerance by moving the cursor to the desired tolerance and pressing **ENTER**. The closest available standard component value will then be calculated, along with the color bands for a resistor of that value, and the results will be displayed in a separate screen.

At any point, press **UP** to return to Miscellaneous Info, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.

Standard Prefixes

Upon selecting this function, a screen containing common prefixes will appear.

Browse through the list to find the desired prefix, or use the search mode. (For more information on using the search mode, refer to “Using the Search Mode” in Chapter 1.) When you have found the desired prefix, press **ENTER** to display the prefix description and value on a full screen, press **→STK** to copy the value to the stack, or press **PRINT** to print the value on an IR printer.

When you have finished browsing the prefix list, press **UP** to return to Miscellaneous Info, **MAIN** to return to the Main menu, or **ATTN** to quit Spice48.

Appendix A

Warranty and Service

Pocket Professional™ Support

You can get answers to your questions about using your Pocket Professional™ Pac from Sparcom. If you don't find the information in this manual or in the HP 48SX Owner's Manual, contact us in one of the following ways:

❶ E-Mail

From Internet: support@sparcom.com
From CompuServe: >Internet:support@sparcom.com
From FidoNet: To:support@sparcom.com

❷ Standard Mail

Sparcom Corporation
897 NW Grant Avenue
Corvallis, OR 97330
Attn: Technical Support Department

❸ Telephone

(503) 757-8416
9 a.m. – 5 p.m. Pacific Standard Time

❹ FAX

(503) 753-7821

Limited One-Year Warranty

What is Covered

A Pocket Professional™ Pac is warranted by Sparcom Corporation against defects in material and workmanship for one year from the date of original purchase. If you sell your card or give it as a gift, the warranty is automatically transferred to the new owner and remains in effect for the original one-year period. During the warranty period, we will repair or replace (at no charge) a product that proves to be defective, provided you return the product and proof of purchase, shipping prepaid, to Sparcom.

What is Not Covered

This warranty does not apply if the product has been damaged by accident or misuse or as the result of service or modification by any entity other than Sparcom Corporation.

No other warranty is given. The repair or replacement of a product is your exclusive remedy. ANY OTHER IMPLIED WARRANTY OF MERCHANTABILITY OR FITNESS IS LIMITED TO THE ONE-YEAR DURATION OF THIS WRITTEN WARRANTY. IN NO EVENT SHALL SPARCOM CORPORATION BE LIABLE FOR CONSEQUENTIAL DAMAGES. Products are sold on the basis of specifications applicable at the time of manufacture. Sparcom shall have no obligation to modify or update products, once sold.

If the Card Requires Service

Sparcom will repair a card, or replace it with the same model or one of equal or better functionality, whether it is under warranty or not.

Service Charge

There is a fixed charge for standard out-of-warranty repairs. This charge is subject to the customer's local sales or value-added tax, wherever applicable. Cards damaged by accident or misuse are not covered by fixed charges. These charges are individually determined based on time and material.

Shipping Instructions

If your card requires service, ship it to the address above and:

- ❶ Include your return address and a description of the problem.
- ❷ Include a proof of purchase date if the warranty has not expired.
- ❸ Include a purchase order, along with a check or credit card number and expiration date (VISA or MasterCard), to cover the standard repair charge.
- ❹ Ship your card, postage prepaid, in protective packaging adequate to prevent damage. Shipping damage is not covered by the warranty, so insuring the shipment is recommended.

Cards are usually serviced and re-shipped within five working days.

Environmental Limits

The reliability of a Pocket Professional™ Pac depends upon the following temperature and humidity limits:

- ❶ Operating Temperature: 0 to 45° C (32 to 113° F).
- ❷ Storage Temperature: -20 to 60° C (-4 to 140° F).
- ❸ Operating and Storage Humidity: 90% relative humidity at 40° C (104° F) maximum.

Appendix B

Summary of Operations

Summary of Screen Softkeys

Screen	Softkeys					
Analysis Manager	MAIN	STK	PRINT	VIEW	FONT	UP
Analysis Parameter	MAIN	STK	PRINT	CALC	FONT	UP
Circuit Manager	ADD	DEL	PACK	XFER	FONT	UP
Constant Library	MAIN	STK	PRINT	UNITS	FONT	UP
Data File Manager	MAIN	DEL		XFER	FONT	UP
Data View	MAIN	SAVE	PLOT	DISP	PREC	UP
	MAIN	STK	PRINT	MEM	FONT	UP
Element Coordinates	MAIN		EDIT	AUTO	CLEAR	UP
Element Manager	ADD	DEL	EDIT			UP
Element Parameter	MAIN	STK	PRINT	VIEW	FONT	UP
Frequency View	MAIN	MARK	PLOT	NODE	FREQ	UP
	MAIN	SAVE	MEM	DISP	PREC	UP
Function Analysis	ROOT	ISECT	SLOPE	AREA	EXTR	EXIT
	F X	F'	NXEQ			
Graphics Environment	ZOOM	Z-BOX	CENT	COORD	LABEL	FCN
	DOT+	DOT-	LINE	TLINE	BOX	FIRCL
	MARK	REPL	SUB	DEL	+ -	KEYS

Graphics Manager	MAIN	→STK	PRINT	VIEW	FONT	UP
Main	ABOUT	→STK	PRINT	VIEW	FONT	QUIT
Miscellaneous Info	MAIN	→STK	PRINT	VIEW	FONT	UP
Node Coordinates	MAIN	_____	EDIT	_____	CLEAR	UP
Node Manager	ADD	DEL	EDIT	REFER	_____	UP
Node Plot	None					
Node View	MAIN	MARK	PLOT	NODE	????	UP
	MAIN	SAVE	MEM	DISP	PREC	UP
Resistor Colors	MAIN	1	2	3	CALC	UP
	MAIN	4	5	PLOT	CALC	UP
Source Value View	MAIN	MARK	PLOT	NODE	SRC	UP
	MAIN	SAVE	MEM	DISP	PREC	UP
Text Editing	←SKIP	SKIP→	←DEL	DEL→	INS	↑STK
Time View	MAIN	MARK	PLOT	NODE	TIME	UP
	MAIN	SAVE	MEM	DISP	PREC	UP
Transient Parameter	MAIN	→STK	PRINT	VIEW	FONT	UP
Value Plot	None					
Wire Coordinates	ADD	DEL	EDIT	AUTO	CLEAR	UP

Summary of Softkey Actions

Key	Action
1	Sets 1st band to the color currently selected by the cursor.
2	Sets 2nd band to the color currently selected by the cursor.
3	Sets 3rd band to the color currently selected by the cursor.
4	Sets 4th band to the color currently selected by the cursor.
5	Sets 5th band to the color currently selected by the cursor.
ABOUT	Displays a screen containing the revision number and product information about the Spice48 Pac.
ADD	Adds a new circuit, element, node, or wire.
AUTO	Automatically generates wire coordinates or element coordinates and orientations from the node coordinates.
CALC	Begins an analysis, using the specified analysis parameters, or calculates the resistor value and tolerance from the colors.
CLEAR	Clears all of the current node, element, or wire coordinates.
COORD	Displays the coordinates of the graphics cursor position.
DEL	Deletes the circuit, element, node, wire, or data file currently selected by the pointer.
DEL→	Deletes all characters in the current word between the cursor's current position and the first character of the next word.
←DEL	Deletes all characters in the current word prior to the cursor.
DISP	Prompts for the display type.
EDIT	Edits the element, node, or coordinates currently selected by the cursor.
F X	Displays the function value at the horizontal axis value of the graphics cursor, and moves the graphics cursor to that point on the function. (Equation plots only.)
F'	Plots the first derivative the current function. (Equation plots only.)
FCN	Displays the Function Analysis menu for further analysis. (Function Analysis menu is useful only for equation plots.)
FONT	Toggles between the small and large fonts.

FREQ	Switches to the Frequency View.
INS	Toggles between insert and type-over modes.
MAIN	Returns to the Main menu.
MARK	Toggles output node, source value, frequency, or time currently selected by the cursor between marked and unmarked states.
MEM	Displays the size of the data slice(s) and the available free memory.
NODE	Switches to the Node View.
NXEQ	Switches to the next function in the value plot. (Equation plots only.)
PACK	Packs the circuit currently selected by the pointer.
PICT	Displays a reference chart indicating the color restrictions.
PLOT	Plots one or more data slices in a node or value plot.
PRES	Prompts for the display precision.
PRINT	Prompts for ONE or ALL to select items, and then sends those items to an IR printer.
QUIT	Quits the Spice48 Pac to the HP 48SX stack.
REFER	Sets the node currently selected by the cursor to be the reference node for the current circuit.
SAVE	Saves one or more data slices.
-SKIP	Moves the cursor to the beginning of the current word.
SKIP-	Moves the cursor to the beginning of the next word.
SLOPE	Displays the slope of the current function at the horizontal axis value of the graphics cursor, and moves the graphics cursor to the point at which the slope was calculated. (Equation plots only.)
SRC	Switches to the Source Value View.
-STK	Prompts for ONE or ALL to select items, and then copies those items to the stack. The items are placed in a list if ALL was chosen.
↑STK	Activates the Interactive Stack, allowing arguments to be copied from the stack to the command line for editing by pressing ECHO .
SUB	Copies the rectangle bounded by the mark and the graphics cursor location to the HP 48SX stack as a graphics object (GROB).

TIME

Switches to the Time View.

UNITS

Pressing this key toggles units, stripping or appending units to all values.

UP

Moves up one level in the menu structure, returning to the previous view, manager, or menu.

VIEW


Displays the entire text of an item too wide to fit on the screen, up to one entire screen size. If the item fits on the screen, this key is non-functional.

XFER

Transfers a circuit or data file to or from a computer.




(Scrolling)

Scrolls plots wider than the screen to view the extra nodes and/or the symbol key. When prefixed with , jumps to the edge of the plot in the indicated direction.



(Graphics)

Moves the graphics cursor in the indicated direction. When prefixed with , moves the graphics cursor to the edge of the screen in the indicated direction.



At the command line, clears text or aborts text entry if the command line is already blank.

At a plot, exits the plot and returns to the previous view.

Elsewhere, quits the Spice48 Pac to the HP 48SX stack.



At a menu, moves down one level in the menu structure, entering the selected function, module, analysis, element, or transient screen.

At the command line, accepts the current command line as the entry and returns to the previous menu or list.

At the Circuit Manager, selects the circuit currently selected by the pointer as the current circuit.

At a parameter screen, prompts for the value of the parameter currently selected by the pointer.

At the Data File Manager, loads the data file currently selected by the pointer.

At the Node View, Source Value View, Frequency View, or Time View, switches to the Data View and displays the data slice of the output node, source value, frequency, or time currently selected by the cursor.

At the resistor color entry screen, calculates the resistor value and tolerance from the colors.



At the Data View or the resistor color results screen, displays the screen title, the item label, and the item, all expanded to one screen.



Toggles between scrolling mode and Graphics Environment. (Graphics Environment is not useful for node plots.)



Dumps the current screen to an IR printer.



If Graphics Environment is active, copies value plot (PICT) to the HP 48SX stack as a graphics object (GROB).

Displays the entire text of an item too wide to fit on the screen, up to one entire screen size. If the item fits on the screen, this key is non-functional.

Appendix C

Index

A

- AC analysis, 95
- AC sweep analysis, 95
- adding
 - a circuit, 63
 - a node, 103
 - an element, 69
- alpha lock, 16
- ammeter, 44
- Analysis Manager
 - analysis parameter entry tips, 90
 - items, 88
 - Summary of Operations, 88
- Analysis Manager, 87
- analysis parameter entry
 - changing values, 91
 - inconsistent values, 91
 - selecting a node, 91
 - selecting an element, 91
 - Summary of Operations, 92
 - tips, 90
- analysis types
 - AC sweep (AC), 95
 - DC sweep (DC), 94
 - operating point (OP), 93
 - transfer function (TF), 93
 - transient sweep (TR), 97
- automatic
 - setting of element coordinates and orientations, 111
 - setting of wire coordinates, 113

B, C

- BJTDC circuit
 - adding circuit, 43
 - adding elements, 44
 - adding nodes, 44
 - ammeter, 44
 - analyzing, 53
 - DC analysis, 55
 - drawing schematic, 50
 - example, 42
 - OP analysis, 53
- capacitor entry, 73
- changing the font size, 14
- Circuit Manager
 - adding a circuit, 63
 - deleting a circuit, 64
 - items, 61
 - packing a circuit, 64
 - Summary of Operations, 62
 - transferring a circuit, 64
- Circuit Manager, 61
- circuit, See Circuit Manager
- clearing all graphics, 115
- clearing the command line, 70
- computer
 - transferring a circuit from, 65
 - transferring a circuit to, 64
 - transferring a data file from, 120
 - transferring a data file to, 119

- Constant Library
 - displaying a value, 151
 - Summary of Operations, 151
- Constant Library, 151
- coordinate entry
 - tips, 108
- copying data from stack, 16
- coupled inductor entry, 75
- current circuit
 - preparing, 89
 - selecting, 63
- current source entry, 77
- current-controlled current source
 - entry, 79
- current-controlled voltage source
 - entry, 81
- Custom Menu, 11

D

- Data File Manager
 - deleting a data file, 119
 - items, 117
 - loading a data file, 119
 - Summary of Operations, 118
 - transferring a data file, 119
- Data File Manager, 117
- data file, see Data File Manager
- data slice
 - definition, 122
 - understanding, 122
- data slice, see node slice, frequency
 - slice, source values slice, time slice
- Data View, see Output Manager
- DC analysis, 94
- DC sweep analysis, 94
- decibels display form, 146
- deleting
 - a circuit, 64
 - a data file, 119
 - a node, 103
 - an element, 69
- directory
 - fileS, 117, 120
 - HOME, 18
 - SPARCOM, 18

- SPICE48D, 18, 63, 66, 90, 117, 120

- Display
 - Precision, 146
 - Type, 145
- Display Forms, 145

E

- editing
 - a node, 103
 - an element, 69
- editing text, 15
- element coordinates and
 - orientations
 - setting, 110
- Element Manager 68
 - adding an element, 69
 - deleting an element, 69
 - editing an element, 69
 - element parameter entry tips, 70
 - items, 68
- Element Manager, 67
- element parameter entry
 - selecting a node, 70
 - selecting an element, 71
 - Summary of Operations, 71
 - tips, 70
- element types
 - capacitor, 73
 - coupled inductors, 75
 - current source, 77
 - current-controlled current
 - source, 79
 - current-controlled voltage
 - source, 81
 - inductor, 74
 - resistor, 72
 - voltage source, 76
 - voltage-controlled current
 - source, 80
 - voltage-controlled voltage
 - source, 78
- element, see Element Manager
- ellipsis, meaning of, 14
- Equation Plot, see Output Manager

- examples
 - BJTDC circuit, 42
 - RC circuit, 19
- exponent softkeys, 70

F

- font size, changing, 14
- frequency slice
 - definition, 123
- Frequency View, see Output Manager
- Function Analysis
 - Summary of Operations, 144
- function transient, 85

G

- GaAs properties, 152
- Graphics Environment
 - BJTDC circuit example, 58
 - Summary of Operations, 144
- Graphics Environment, 144
- Graphics Manager
 - clearing all graphics, 115
 - coordinate entry tips, 108
 - creating an accurate schematic, 107
 - grid coordinates, 107
 - items, 106
 - oversize schematics, 115
 - printing the schematic, 115
 - setting element coordinates and orientations
 - automatic generation, 111
 - Summary of Operations, 111
 - setting element coordinates and orientations, 110
 - setting node coordinates
 - Summary of Operations, 109
 - setting node coordinates, 108
 - setting wire coordinates
 - automatic generation, 113
 - Summary of Operations, 113
 - setting wire coordinates, 112

- Summary of Operations, 106
 - updating the schematic, 115
 - viewing the schematic, 114
- Graphics Manager, 105
- grid coordinates, 107

H, I, J, K, L

- imaginary display form, 146
- inconsistent values, meaning of, 91
- inductor entry, 74
- input screens, 15
- installing a ROM card, 10
- Kermit, 65, 120
- Line Plot, see Output Manager
- loading
 - a data file, 119
- loading data from stack, 16

M

- Macintosh, see computer
- magnitude display form, 146
- Main Menu
 - items, 12
 - Summary of Operations, 13
- Main Menu, 12
- memory
 - estimates, 17
 - free, 17
 - requirements, 17
 - warnings, 18
- Miscellaneous Info
 - items, 148
 - Summary of Operations, 148
- Miscellaneous Info, 147
- moving around the screen, 12

N

- node coordinates
 - setting, 108
- Node Manager
 - adding a node, 103
 - deleting a node, 103
 - editing a node, 103
 - items, 101
 - setting the reference node, 104

- Summary of Operations, 102
- Node Manager, 101
- Node Plot, see Output Manager
- node slice
 - definition, 122
- Node View, see Output Manager

O

- OP analysis, 93
- operation point analysis, 93
- orientation of elements, 111
- Output Manager
 - Data View
 - exiting, 126
 - plotting the data slice, 126
 - saving the data slice, 126
 - Summary of Operations, 127
 - understanding, 124
 - Data View, 123
 - Display Forms
 - Display Precision, 146
 - Display Type, 145
 - Display Forms, 145
 - Frequency View
 - exiting, 135
 - marking frequencies, 134
 - plotting frequency slice(s), 135
 - saving frequency slice(s), 135
 - Summary of Operations, 136
 - using, 134
 - viewing a frequency slice, 135
 - Frequency View, 134
 - Node Plot
 - plotting ranges, 140
 - Summary of Operations, 141
 - Node Plot, 140
 - Node View
 - exiting, 130
 - marking nodes, 128
 - plotting node slice(s), 129
 - saving node slice(s), 129

- Summary of Operations, 130
 - using, 128
 - viewing a node slice, 129
- Node View, 128
- Source Value View
 - exiting, 133
 - marking source values, 131
 - plotting source value slice(s), 132
 - saving source value slice(s), 132
 - Summary of Operations, 133
 - using, 131
 - viewing a source value slice, 132
- Source Value View, 131
- summary of views, 123
- Time View
 - exiting, 138
 - marking times, 137
 - plotting time slice(s), 138
 - saving time slice(s), 138
 - Summary of Operations, 139
 - using, 137
 - viewing a time slice, 138
- Time View, 137
- Value Plot
 - Equation Plot, 143
 - Line Plot, 143
 - plotting ranges, 142
 - Point Plot, 142
 - Summary of Operations, 144
- Value Plot, 141
- Output Manager, 121
- output nodes
 - definition, 122

P, Q

- packing a circuit, 64
- PC, see computer
- phase angle display form, 146
- piecewise-linear transient, 84

- Plots see Output Manager
- plotting ranges, see Output Manager
- Point Plot, see Output Manager
- Precision, 145, 146
- preparing the current circuit, 89
- printing the schematic, 115
- properties
 - GaAs, 152
 - Si, 152
- pulse transient, 83
- PWL transient, see piecewise-linear transient

R

- RAM, see memory
- RC circuit
 - AC analysis, 36
 - adding circuit, 20
 - adding elements, 22
 - adding nodes, 21
 - analyzing, 33
 - drawing schematic, 30
 - example, 19
 - OP analysis, 33
 - TF analysis, 36
 - TR analysis, 39
- real display form, 146
- reference node, setting, 104
- removing a ROM card, 10
- resistor bands
 - descriptions, 149
 - digit bands, 149
 - exponent band, 150
 - restrictions, 149
 - tolerance band, 150
- resistor colors
 - Summary of Operations, 150
- resistor colors, 149
- resistor entry, 72
- ROM card
 - environmental limits, 157
 - installing, 10
 - removing, 10
 - service, 156

S

- schematic, see Graphics Manager

- Search Mode, 14
- selecting the current circuit, 63
- setting the reference node, 104
- Si properties, 152
- slice, data, see data slice
- slice, node, see node slice
- slice, source value, see source value slice
- slice, time, see time slice
- source value slice
 - definition, 123
- Source Value View, see Output Manager
- Sparcom
 - how to contact, 155
- Spice48
 - revision, 11
 - starting, 11
 - tutorial, 19
- stack
 - loading data from, 16
 - standard components, 152
 - standard display form, 146
 - standard prefixes, 153
 - starting Spice48, 11
 - Summary of Operations, 159
 - summary of screen softkeys, 159
 - summary of softkey actions, 161
 - support, 155
 - system flags
 - flag -60 (alpha), 16
 - system flags, 16

T

- technical support, 155
- Text Editing
 - Summary of Operations, 15
- text editing, 15
- TF analysis, 93
- time slice
 - definition, 123
- Time View, see Output Manager
- TR analysis, 97
- transfer function analysis, 93
- transferring
 - a circuit, 64
 - a data file, 119

- transient parameter entry
 - Summary of Operations, 71
- transient sources
 - function transient, 85
 - piecewise-linear transient, 84
 - pulse transient, 83
- transient sources, 82
- transient sweep analysis
 - continuing, 99
 - new, 97
- transient sweep analysis, 97
- tutorial, 19
- types of analysis, see analysis types
- types of elements, see element types

U, V, W, X, Y, Z

- updating the schematic, 115
- user flags
 - flag 57 (font), 17
 - flag 61 (units), 17
- user flags, 17
- Value Plot, see Output Manager
- viewing items too wide for the display, 14
- viewing the schematic, 114
- voltage source entry, 76
- voltage-controlled current source
 - entry, 80
- voltage-controlled voltage source
 - entry, 78
- warranty, 156
- wire coordinates
 - setting, 112