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
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 Manager

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

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

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:

1. Turn the HP 48SX off. Do not press [ON] until you have completed the installation procedure.

2. 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:

3. 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:

4. 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.

5. Replace the port cover.

Removing a ROM Card

To remove a ROM card, follow these steps:
1. Turn the HP 48SX off. Do not press ON until you have completed the removal procedure.

2. 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.

3. Press against the card's grip and slide the card out of the port, as shown below:

4. 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 & [BROWSE] 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:

```
| HOME |
| 4: |
| 3: |
| 2: |
| 1: |
| SPICE [EXIT] |
```

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 [.getLeft] [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:

```
SPICE48
| CIRCUIT MANAGER
| ELEMENT MANAGER
| ANALYSIS MANAGER
| NODE MANAGER
| GRAPHICS MANAGER
| DATA FILE MANAGER
| MISCELLANEOUS INFO
| ABOUT | TOOL | VIEW | FONT | QUIT |
```

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 ENTER.

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.

<table>
<thead>
<tr>
<th>Item</th>
<th>Description</th>
<th>See...</th>
</tr>
</thead>
<tbody>
<tr>
<td>Circuit Manager</td>
<td>Adds, deletes, packs, and transfers circuit data; selects the current circuit; installs tutorial circuit data.</td>
<td>Chapter 3</td>
</tr>
<tr>
<td>Element Manager</td>
<td>Adds, deletes, and edits circuit elements.</td>
<td>Chapter 4</td>
</tr>
<tr>
<td>Analysis Manager</td>
<td>Performs calculations.</td>
<td>Chapter 5</td>
</tr>
<tr>
<td>Node Manager</td>
<td>Adds, deletes, and edits nodes; sets the reference node.</td>
<td>Chapter 6</td>
</tr>
<tr>
<td>Graphics Manager</td>
<td>Adds, deletes, and edits coordinates and creates the schematic.</td>
<td>Chapter 7</td>
</tr>
</tbody>
</table>
### Data File Manager
Loads, deletes, and transfers stored data files.

### Miscellaneous Info
Reference data tables; resistor color chart calculations; standard component values; ASCII table.

---

**Summary of Operations**

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Main</td>
<td>ABOUT -STK PRINT VIEW FONT QUIT</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABOUT</td>
<td>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.</td>
</tr>
<tr>
<td>FONT</td>
<td>Toggles between the small and large fonts.</td>
</tr>
<tr>
<td>PRINT</td>
<td>Prompts for ONE or ALL to select items, and then sends those items to an IR printer.</td>
</tr>
<tr>
<td>QUIT</td>
<td>Quits the Spice48 Pac to the HP 48SX stack.</td>
</tr>
<tr>
<td>-STK</td>
<td>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.</td>
</tr>
<tr>
<td>VIEW</td>
<td>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.</td>
</tr>
<tr>
<td>ALT</td>
<td>Quits the Spice48 Pac to the HP 48SX stack.</td>
</tr>
<tr>
<td>ENTER</td>
<td>Moves down one level in the menu structure, entering the module selected by the pointer.</td>
</tr>
<tr>
<td>ON</td>
<td>Dumps the current screen to an IR printer.</td>
</tr>
</tbody>
</table>

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

The default font for the Spiced8 Pac displays information in condensed, uppercase letters only. Pressing FONT will toggle the information to a larger font, which is case-sensitive:

```
SPICE48
– Circuit Manager
  Element Manager
  Analysis Manager
  Node Manager
  Graphics Manager
  Data File Manager
```

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 EXIT to return to the menu. At all screens, including those screens where VIEW is not present, pressing [→ VST] will perform the same function. If an item does fit entirely on the screen, VIEW or [← VST] 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 [α] to display the following screen:

```
| HOME | SPARC | SPICE4BD |
Search for:  |
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SKIP</td>
<td>SKIP</td>
</tr>
<tr>
<td></td>
<td>DEL</td>
<td>DEL+</td>
</tr>
<tr>
<td></td>
<td>INS</td>
<td></td>
</tr>
</tbody>
</table>
```

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 **[ATN]**.

---

**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

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Text Editing</td>
<td>←SKIP SKIP± DEL DEL= INS †STK</td>
</tr>
</tbody>
</table>

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

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 \(\alpha\) places the HP 48SX in alpha-entry mode for only one character, and you must press \(\alpha\alpha\) to lock alpha-entry mode. If flag -60 is set, however, then pressing \(\alpha\) only once locks alpha-entry mode. The examples in this manual assume that flag -60 is clear, so that each \(\alpha\) 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 \(\alpha\), or if that does not work, press \(\alpha\alpha\) to display the EDIT menu and then press \(\alpha\alpha\alpha\). 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 \(\alpha\) and \(\alpha\), and when you reach the desired value, press \(\alpha\) to copy it to the command line for editing. To exit the Interactive Stack and return to the command line, press \(\alpha\alpha\) or \(\alpha\alpha\alpha\). 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 \(\alpha\alpha\alpha\) 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 " " or " " 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 " " 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 " " presses, but it cannot properly restore your stack and flag settings if you push " " 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:

1. Press $\text{LIB}$ to display all libraries available to your HP 48SX.
2. Find and press $\text{SPICE}$ to enter the Spice48 Pac library directory.
3. 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:

1. Add the circuit RC.
2. Add nodes to RC.
3. Add elements to RC.
4. Draw the schematic of RC.
5. 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**:

![Pick Circuit screen](image)

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

![Enter circuit name screen](image)
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:](image)

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:

![Node Manager](image)

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

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

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:

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**:

### Adding Element VS

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

Press **ENTER** to accept the element type and 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.):

```
PRG  WDME SPARCOM SPICE4BD RC
Enter element name:
+ name
VS
```

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:

```
V-Source
ELEMENT: VS
+ NODE (+): PRESS ENTER
NODE (-): PRESS ENTER
DC VALUE (V): 0
AC VALUE (V): (0.0)
TRANSIENT: NONE
```

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 , to move the cursor to node 1:

```
Pick Node (+):

[Refer] 1 | 2
```

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:

```
V-Source
ELEMENT: VS
+ NODE (+): 1
NODE (-): PRESS ENTER
DC VALUE (V): 0
AC VALUE (V): (0.0)
TRANSIENT: NONE
```
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](image)

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 **ASN** or **C** 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](image)

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 **ASN** 1 **<DEL** 0, to set the AC value to (1,0):

```
Edit AC Value (V):
+ cmplx (m, a) or (r, i)
(1+i)
```

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:
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:

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 μsec, 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 μsec, press MICRO after the value at the prompt):

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:

This completes Step 30. Press [UP] to return to the Element Manager:
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”:

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

**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:
Entering the (+) Node of $R_1$

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
```

Entering the (-) Node of $R_1$

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
```

Entering the Value of $R_1$

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
```

This completes Step 3. Press [UP] to return to the Element Manager:
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 parameter screen](image)

**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 5 ▼️, to set the value to −5 V. Press ENTER to accept the value and return to the element parameter screen:
This completes Step 33. Press UP to return to the Element Manager:

<table>
<thead>
<tr>
<th>Elements:</th>
</tr>
</thead>
<tbody>
<tr>
<td>VS</td>
</tr>
</tbody>
</table>

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:

<table>
<thead>
<tr>
<th>Graphics Manager</th>
</tr>
</thead>
<tbody>
<tr>
<td>→ SET NODE COORDINATES</td>
</tr>
<tr>
<td>SET ELMT COORDINATES</td>
</tr>
<tr>
<td>SET WIRE COORDINATES</td>
</tr>
<tr>
<td>VIEW SCHEMATIC</td>
</tr>
<tr>
<td>PRINT SCHEMATIC</td>
</tr>
<tr>
<td>CLEAR ALL GRAPHICS</td>
</tr>
</tbody>
</table>

**Setting Node Coordinates**

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

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

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:

This completes Step 43. 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:

\[
\text{Elements:} \quad (0.1,1) \text{ VS } (0.3,1) \quad R1 \quad (1,1,1) \text{ CBU} 
\]

This completes Step 4. 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:

\[
\text{Wires:} 
\]

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:

\[
\text{Wires:} \quad (0,0) \quad (1,0) 
\]

This completes Step 5. 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 4 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 CAL. 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:
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 ALT to return to the Output Manager Data View.

Checking Free Memory

Press mem to determine the free memory required to save the OP analysis data:

Press ENTER or ALT to return to the Output Manager Data View.

Saving the Results

If you have enough free memory, press 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 3. 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 Screen]

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 Screen]

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 5. 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:

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):

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):

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 \[ \text{ATN} \] 20 \[ \text{Y-} \] 0 ENTER to adjust the vertical range slightly. The following plot appears:

![dB vs. Frequency Plot](image)

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 \[ \text{ATN} \] to return to the Output Manager Data View, and press \[ \text{UP} \] to return to the Output Manager Node View.

Changing Display Type

Press \[ \text{ATN} \] DISP [LEFT] 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 \[ \text{ATN} \] PLOT to create a phase vs. frequency Bode plot, then [LEFT] ENTER to select a point plot, then ENTER to accept the default horizontal range, then \[ \text{ATN} \] 90 \[ \text{Y-} \] 0 ENTER to adjust the vertical range slightly. The following plot appears:

![Phase vs. Frequency Plot](image)

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

When you have finished viewing the plot, press \[ \text{ATN} \] to return to the Output Manager Node View.
Saving the Results

Press \[ \text{Nat MEM} \] to determine the free memory required to save the AC analysis data. If you have enough free memory, press \[ \text{SAVE} \], and at the prompt press \[ \text{ENTER} \] to accept the default name. The AC analysis data will be stored into the file 'RC.AC' in the 'files' subdirectory of the 'SPICE4SD' directory. (For more information, refer to Chapter 8, "Data File Manager.")

This completes Step 5. Press \[ \text{UP} \] 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 \( C_{OUT} \) 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 \[ \text{ENTER} \]. The analysis parameter screen appears:

![Transist Sweep Parameter Screen]

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 \[ \text{CALC} \]. 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 Node View]

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.):

<table>
<thead>
<tr>
<th>TR: Times</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0000</td>
</tr>
<tr>
<td>0.2000</td>
</tr>
<tr>
<td>0.4000</td>
</tr>
<tr>
<td>0.6000</td>
</tr>
<tr>
<td>0.8000</td>
</tr>
<tr>
<td>1.0000</td>
</tr>
<tr>
<td>1.2000</td>
</tr>
<tr>
<td>1.4000</td>
</tr>
<tr>
<td>1.6000</td>
</tr>
<tr>
<td>1.8000</td>
</tr>
<tr>
<td>2.0000</td>
</tr>
<tr>
<td>2.2000</td>
</tr>
<tr>
<td>2.4000</td>
</tr>
<tr>
<td>2.6000</td>
</tr>
<tr>
<td>2.8000</td>
</tr>
</tbody>
</table>

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.):

<table>
<thead>
<tr>
<th>TR: Time=2.0000</th>
</tr>
</thead>
<tbody>
<tr>
<td>V(k1): 0.0000</td>
</tr>
<tr>
<td>V(k2): 2.1000</td>
</tr>
<tr>
<td>IC(k1): 2.1000</td>
</tr>
</tbody>
</table>

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 **MARK** to change the display precision to standard. (For more information, refer to “Display Forms” in Chapter 9.)

Marking All Data

Press **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 **<** 12 **>** 10 **ENTER** to adjust the vertical range slightly. The following plot (and symbol key) appears:
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 X 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 [ Exit ] to return to the Output Manager Node View.

**Saving the Results**

Press [ Exit ] [ 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 68. 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, \( \beta_{DC} \), according to

\[
\beta_{DC} = \frac{I_C}{I_B},
\]

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.) \( \beta_{DC} \) 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 large 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 \( V(COLLECTOR) - V(EMITTER) \) in the analyses below.)
To investigate this circuit, you will follow these steps:

1. Add the circuit BJTDC.
2. Add elements and nodes to BJTDC.
3. Draw the schematic of BJTDC.
4. 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:](image)

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:](image)

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 1. 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:

![BJTDC Diagram]

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 EMMITTER.
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>:

```
<table>
<thead>
<tr>
<th>Element</th>
</tr>
</thead>
<tbody>
<tr>
<td>PRESS ADD</td>
</tr>
</tbody>
</table>
```

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:

```
<table>
<thead>
<tr>
<th>Pick Node (+):</th>
</tr>
</thead>
<tbody>
<tr>
<td>PRESS ADD</td>
</tr>
</tbody>
</table>
```

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:

```
<table>
<thead>
<tr>
<th>Pick Node (+):</th>
</tr>
</thead>
<tbody>
<tr>
<td>SOURCE</td>
</tr>
</tbody>
</table>
```

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 \([\text{Enter}]\) to select the \((-\) node of this element, although it does not yet exist in circuit BJTDC. Press \([\text{ADD}]\) to add a new node, and at the prompt type GROUND, the name of the \((-\) node for VS. When you have finished, press \([\text{Enter}]\) to accept the name and return to the Node Manager. The new node is now listed. Press \([\text{Enter}]\) 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 \((2)\). Press \([\text{Menu}]\) 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 \([\text{ADD}]\) to add a new element, leave the cursor at "R—Resistor", and press \([\text{Enter}]\) 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
\[\text{ELEMENT: RS}
\[\text{NODE (+): SOURCE}
\[\text{NODE (-): INPUT}
\[\text{VALUE (R): 1000}
```

This completes Step \((2)\). Press \([\text{Menu}]\) to return to the Element Manager.

Adding CS and Node BASE

Press \([\text{ADD}]\) to add a new element, move the cursor to "C—Capacitor", and press \([\text{Enter}]\) 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:
Adding RB1, RB2 and Node BATTERY

Press \textbf{ADD} to add a new element, leave the cursor at “R–Resistor”, and press \textbf{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:

\begin{verbatim}
R-Resistor
\textbf{ELEMENT: RB1}
\textbf{NODE (+): BATTERY}
\textbf{NODE (-): BASE}
\textbf{VALUE (R): 54000}
\end{verbatim}

Press \textbf{UP} to return to the Element Manager.

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

\begin{verbatim}
R-Resistor
\textbf{ELEMENT: RB2}
\textbf{NODE (+): BASE}
\textbf{NODE (-): GROUND}
\textbf{VALUE (R): 20000}
\end{verbatim}

This completes Step 2. Press \textbf{UP} to return to the Element Manager.

Adding VBE, RE, CE, and Node EMITTER

Press \textbf{ADD} to add a new element, move the cursor to “V–Source”, and press \textbf{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:
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 (R): 5000
```

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
```

This completes Step 2. 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
```
This completes Step 2⑧. 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:

<table>
<thead>
<tr>
<th>V-Source</th>
</tr>
</thead>
<tbody>
<tr>
<td>ELEMENT: VAMTR</td>
</tr>
<tr>
<td>NODE (+): COLLECTOR</td>
</tr>
<tr>
<td>NODE (-): AMMETER</td>
</tr>
<tr>
<td>DC VALUE (V): 0</td>
</tr>
<tr>
<td>AC VALUE (V): (0,0)</td>
</tr>
<tr>
<td>TRANSIENT: NONE</td>
</tr>
</tbody>
</table>

This completes Step 2⑧. 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:

<table>
<thead>
<tr>
<th>R-Resistor</th>
</tr>
</thead>
<tbody>
<tr>
<td>ELEMENT: RC</td>
</tr>
<tr>
<td>NODE (+): BATTERY</td>
</tr>
<tr>
<td>NODE (-): COLLECTOR</td>
</tr>
<tr>
<td>VALUE (R): 10000</td>
</tr>
</tbody>
</table>

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:

<table>
<thead>
<tr>
<th>R-Resistor</th>
</tr>
</thead>
<tbody>
<tr>
<td>ELEMENT: RL</td>
</tr>
<tr>
<td>NODE (+): OUTPUT</td>
</tr>
<tr>
<td>NODE (-): GROUND</td>
</tr>
<tr>
<td>VALUE (R): 1000</td>
</tr>
</tbody>
</table>

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): 0.000001
   INITIAL (V): 0
```

This completes Step 2(4). 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): 0
   AC VALUE (V): 0
   TRANSIENT: NONE
```

This completes Step 2(5). 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.
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:

<table>
<thead>
<tr>
<th>Nodes:</th>
</tr>
</thead>
<tbody>
<tr>
<td>(0.1) SOURCE (0.0) GND (0.2) INPUT</td>
</tr>
<tr>
<td>(1.2) EMSE (6.3) BATTE (2.1) EMITT</td>
</tr>
<tr>
<td>(3.2) HNME (4.2) COLLE (5.2) OUTPUT</td>
</tr>
</tbody>
</table>

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:

<table>
<thead>
<tr>
<th>Elements:</th>
</tr>
</thead>
<tbody>
<tr>
<td>(0.1) VCC (0.1) LS (0.2) CS</td>
</tr>
<tr>
<td>(6.3) R1 (1.2) L1 (1.2) B VEE</td>
</tr>
<tr>
<td>(2.1) CE (2.1) CE (3.2) FCE</td>
</tr>
<tr>
<td>(4.2) VMH (6.2) R2 (5.2) RL</td>
</tr>
<tr>
<td>(4.2) CL (6.3) VMH</td>
</tr>
</tbody>
</table>

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 \[\text{ENTER}\] to return to the Graphics Manager.

This completes Step 33.

Adjusting Element Coordinates and Orientations

Move the pointer to “Set Elnt Coordinates” and press \[\text{ENTER}\]. The circuit elements will be displayed. Adjust the coordinates and orientations of elements RB1, CE, and RC, until your screen appears as follows:

<table>
<thead>
<tr>
<th>Elements:</th>
</tr>
</thead>
<tbody>
<tr>
<td>(0.1, 1 US)</td>
</tr>
<tr>
<td>(1.2, 1) RE</td>
</tr>
<tr>
<td>(2.1, 1 LE)</td>
</tr>
<tr>
<td>(4.2, 2 CRM)</td>
</tr>
</tbody>
</table>

This completes Step 34. Press \[\text{UP}\] to return to the Graphics Manager.

Setting Wire Coordinates

Move the pointer to “Set Wire Coordinates” and press \[\text{ENTER}\]. 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:

<table>
<thead>
<tr>
<th>Wires:</th>
</tr>
</thead>
<tbody>
<tr>
<td>(0.0, 5.0)</td>
</tr>
<tr>
<td>(0.0, 3.0)</td>
</tr>
<tr>
<td>(1.1, 1.0)</td>
</tr>
<tr>
<td>(5.1, 5.0)</td>
</tr>
</tbody>
</table>

This completes Step 35. Press \[\text{UP}\] to return to the Graphics Manager.

Viewing the Finished Schematic

Move the pointer to “View Schematic” and press \[\text{ENTER}\]. You will be prompted to update the schematic. Press \[\text{YES}\] 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 3© 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 thereon the collector current (a sensitivity parameter: $\frac{\partial I}{\partial VBE}$). 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.

![Operating Point](image)

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:

![OP: Freq=0](image)

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.):

![Node Plot](image)

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

When you have finished viewing the plot, press \[ \text{Esc} \] to return to the Output Manager Data View.

**Saving the Results**

Press \[ \text{M} \] \[ \text{M} \] \[ \text{MEM} \] to determine the free memory required to save the OP analysis data. If you have enough free memory, press \[ \text{M} \] \[ \text{S} \] \[ \text{SAVE} \], and at the prompt press \[ \text{Enter} \] to accept the default name. The OP analysis data will be stored into the file 'BJTDC.OP' in the 'files' subdirectory of the 'SPICE4SD' directory.

**Further OP Analysis**

If you decrease the value of resistor RB2 from 20K\(\Omega\) down to 1.5K\(\Omega\) 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\(\Omega\) up to 54K\(\Omega\) and perform an OP analysis, you will see a saturation case where V(COLLECTOR) - V(EMITTER) < 0. In an actual transistor, this value would be 0.

This completes Step 4\(\circ\). Press \[ \text{UP} \] 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(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 \[ \text{Enter} \]. The analysis parameter screen appears:

```
DC Sweep
\-SOURCE: PRESS ENTER
START (V,R): PRESS ENTER
STOP (V,R): PRESS ENTER
STEP (V,R): PRESS ENTER
MEMORY: ???/18974.5
\-MAIN | \-STEP | \-PRINT | \-CALC | \-POINT | \-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 \[ \text{CALC} \]. 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.):

<table>
<thead>
<tr>
<th>DC: Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>V(SOURCE)</td>
</tr>
<tr>
<td>V(BATTERY)</td>
</tr>
<tr>
<td>V(Collector)</td>
</tr>
<tr>
<td>I(BE)</td>
</tr>
</tbody>
</table>

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.):

<table>
<thead>
<tr>
<th>DC: VBE Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
</tr>
<tr>
<td>.675</td>
</tr>
<tr>
<td>.75</td>
</tr>
</tbody>
</table>

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 EXIT 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.):

<table>
<thead>
<tr>
<th>DC: VBE=.65</th>
</tr>
</thead>
<tbody>
<tr>
<td>V(SOURCE): 0</td>
</tr>
<tr>
<td>V(INPUT): 0</td>
</tr>
<tr>
<td>V(BASE): 5.27103355006</td>
</tr>
<tr>
<td>V(BATTERY): 20</td>
</tr>
<tr>
<td>V(EMITTER): 4.62103355006</td>
</tr>
<tr>
<td>V(AMMETER): 10.8478543563</td>
</tr>
<tr>
<td>V(COLLECTOR): 10.8478543563</td>
</tr>
<tr>
<td>V(OUTPUT): 0</td>
</tr>
</tbody>
</table>

This is identical to the earlier OP analysis data.

56 2: Spice Tutorial
When you have finished viewing the data, press \textbf{UP} to return to the Output Manager Source Value View and press \textbf{NODE} to switch to the Output Manager Node View.

### Plotting $I(V_{AMTR})$ vs. $V(V_{BE})$ (Value Plot, Equation)

Move the cursor to the output node $I(V_{AMTR})$ and press \textbf{PLOT}, then \textbf{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 \textbf{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 `0 0.00075 \textbf{PC} 0.001`:

```
{ HOME SPARCOM SPICE4BD BJTC3
Edit vertical range:
@ min max
(reals, min<max )
.00075 .001
```

Press \textbf{ENTER} to accept the vertical range. Next, you will be prompted to edit the plot resolution:

```
{ HOME SPARCOM SPICE4BD BJTC3
Edit plot resolution:
@ #binary for pixels
@ real for user-units
#1
```

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 \textbf{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 ~2 \(\mu\text{A}\). This is the sensitivity parameter \(\frac{\partial I}{\partial VBE}\). (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 \[^{MEM}\] 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:

1. Press \(\text{LIB}\) to display all libraries available to your HP 48SX.
2. Find and press \(\text{SPICE}\) to enter the Spice48 Pac library directory.
3. Press the first softkey, \(\text{SPICE}\), to start the Spice48 Pac.
4. At the Main menu, make sure the pointer is at “Circuit Manager” and press \(\text{ENTER}\).

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**

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Circuit Manager</td>
<td>ADD, DEL, PACK, XFER, FONT, UP</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>ADD</td>
<td>Prompts for the name of a new circuit and initializes it.</td>
</tr>
<tr>
<td>DEL</td>
<td>Deletes the circuit currently selected by the pointer.</td>
</tr>
<tr>
<td>FONT</td>
<td>Toggles between the small and large fonts.</td>
</tr>
<tr>
<td>PACK</td>
<td>Packs the circuit currently selected by the pointer.</td>
</tr>
<tr>
<td>XFER</td>
<td>Transfers a circuit to or from a computer.</td>
</tr>
<tr>
<td>UP</td>
<td>Moves up one level in the menu structure, returning to the Main menu.</td>
</tr>
<tr>
<td>EXIT</td>
<td>Quits the Spice48 Pac to the HP 48SX stack.</td>
</tr>
<tr>
<td>ENTER</td>
<td>Selects the circuit currently selected by the pointer as the current circuit for Spice48 Pac operations.</td>
</tr>
<tr>
<td>ON-PREV</td>
<td>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.</td>
</tr>
<tr>
<td>OFF-PREV</td>
<td>Dumps the current screen to an IR printer.</td>
</tr>
</tbody>
</table>
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:

```
< HOME SPARCOM SPICE48D >
Enter circuit name:

Type in a name for the new circuit and press **ENTER**, or to abort the add operation, press **DEL** to clear the command line (if necessary), and then press **ENTER** or **EXIT** 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 \[ \text{DEL} \]. You will be prompted to verify the delete operation by pressing \[ \text{YES} \] to confirm or \[ \text{NO} \] or \[ \text{ABORT} \] 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 \[ \text{PACK} \].

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 \[ \text{XFER} \].
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 **EXIT**. 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 **F3**, and press **F9**. 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 EXIT. 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:

**Pick Circuit:**
- FREE MEMORY: 15190.5
- JUST DOWNLOADED: 582.0 (CURRENT)
- BJTDC: 6838.0
- RC: 2244.5
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:

1. Press [LIB] 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.
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.”):

<table>
<thead>
<tr>
<th>Elements:</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>V5</strong></td>
</tr>
<tr>
<td><strong>R5</strong></td>
</tr>
<tr>
<td><strong>RE</strong></td>
</tr>
<tr>
<td><strong>VWHITE</strong></td>
</tr>
<tr>
<td><strong>CL</strong></td>
</tr>
<tr>
<td><strong>ADD</strong></td>
</tr>
</tbody>
</table>

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

**Summary of Operations**

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element Manager</td>
<td>ADD  DEL  EDIT  UP</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>ADD</strong></td>
<td>Adds a new element to the current circuit.</td>
</tr>
<tr>
<td><strong>DEL</strong></td>
<td>Deletes the element currently selected by the cursor.</td>
</tr>
<tr>
<td><strong>EDIT</strong></td>
<td>Edits the element currently selected by the cursor.</td>
</tr>
<tr>
<td><strong>UP</strong></td>
<td>Moves up one level in the menu structure, returning to the Main menu.</td>
</tr>
<tr>
<td><strong>EXIT</strong></td>
<td>Quits the Spice48 Pac to the HP 48SX stack.</td>
</tr>
<tr>
<td><strong>ENTER</strong></td>
<td>Edits the element currently selected by the cursor, or if no elements exist, adds a new element to the current circuit</td>
</tr>
<tr>
<td><strong>ON-REV</strong></td>
<td>Dumps the current screen to an IR printer.</td>
</tr>
</tbody>
</table>

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-VSS | F-CCVS | G-CCVS |
```

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 EXIT 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 EXIT 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 EDIT 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 EXIT to clear the command line (if necessary) and then
press \texttt{ENTER} or \texttt{DEL} to return to the element parameter screen without changing
the parameter value. When you have finished editing the current element, press
\texttt{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.

\section*{Element Parameter Entry Tips}

Here are some useful things to know about entering or editing element param-
ters:

- To enter or edit the value of a parameter, move the pointer to it and
  press \texttt{ENTER}. After entering or editing the value of a parameter, press
  \texttt{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 \texttt{DEL} to clear the command line (if necessary) and then
  press \texttt{ENTER} or \texttt{DEL} to return to the analysis parameter screen without
  changing the parameter value.

- If you press \texttt{UP}, \texttt{MAIN}, or \texttt{FIN} while the new element is only par-
tially specified, you will be asked to verify that you want to abandon
the partial entry. Press \texttt{YES} to abandon the entry, or \texttt{NO} or \texttt{FIN}
to continue specifying the new element.

- You only need to enter values for those parameters that initially dis-
play, "Press ENTER.” Other parameters (e.g., the initial value of a ca-
pacitor, the transient type of a voltage source) will be given default val-
ues (e.g., 0 and \texttt{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
\texttt{MICRO} to enter the value \texttt{.000003}). More exponent softkeys can be
accessed by pressing \texttt{\textasciicircum}

\section*{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 \texttt{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 Exit 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 Exit 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

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element/Transient Parameter</td>
<td>Main, Exit, Print, View, Font, Up</td>
</tr>
</tbody>
</table>
### 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.
**ENT** | Quits the Spice48 Pac to the HP 48SX stack. The element is saved automatically.
**ON-PREV** | Prompts for the value of the parameter currently selected by the pointer.

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:

```plaintext
R-Resistor

>ELEMENT: PRESS ENTER
NODE (+): PRESS ENTER
NODE (-): PRESS ENTER
VALUE (R): PRESS ENTER
```

---

72 4: Element Manager
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 Parameter Screen]

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:

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:

$$M = k \sqrt{L_1 L_2}$$

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:

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:

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:

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 "1" and include no spaces. Possible names include: "12," "13amps," "1SRC," 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,phase).

- **Transient**: Selected from NONE, PULSE, PWL, and FUNCTION, defaults to NONE. (For more information, refer below to "Transient Sources.")
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 \textbf{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:

\[
V(+) - V(-) = Gain \left[ V(C(+)) - V(C(-)) \right].
\]
F-CCCS (Current-Controlled Current Source)

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

![F-CCCS Parameter Screen]

You should now enter or edit the parameter values. When you have finished, press \[ \text{UP} \] 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:

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

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:

You should now enter or edit the parameter values. When you have finished, press \text{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:

\[
I(\text{Source}) = \text{Transcon} \left[ V(C(+)) - V(C(−)) \right].
\]
H-CCVS (Current-Controlled Voltage Source)

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

![Parameter Screen]

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:

\[ V(\text{Source}) = Transres \cdot I(\text{Controller}) \]

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

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 EXIT to quit Spice4.)

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:

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 Screen](image)

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 **Vdc** is the DC Value of the source:

![PWL Transient Diagram](image)

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 9.5 E 499. A non-periodic transient should have a value of 0 or 9.5 E 499 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:

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 $t$ 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 9.5 E 499. A non-periodic transient should have a value of 0 or 9.5 E 499 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:

1. Press \[ \text{LIBRARY} \] to display all libraries available to your HP 48SX.
2. Find and press \[ \text{SPICE} \] to enter the Spice48 Pac library directory.
3. Press the first softkey, \[ \text{SPICE} \], to start the Spice48 Pac.
4. At the Main menu, make sure the pointer is at “Analysis Manager” and press \[ \text{ENTER} \]:

![Analysis Manager Menu]

5: Analysis Manager 87
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 \textit{ENTER}.

\textbf{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.

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

\textbf{Summary of Operations}

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analysis Manager</td>
<td>MAIN \quad \texttt{-STK} \quad PRINT \quad VIEW \quad FONT \quad UP</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>\texttt{FONT}</td>
<td>Toggles between the small and large fonts.</td>
</tr>
<tr>
<td>\texttt{MAIN}</td>
<td>Returns to the Main menu.</td>
</tr>
<tr>
<td>\texttt{PRINT}</td>
<td>Prompts for \texttt{ONE} or \texttt{ALL} to select items, and then sends those items to an IR printer.</td>
</tr>
<tr>
<td>\texttt{-STK}</td>
<td>Prompts for \texttt{ONE} or \texttt{ALL} to select items, and then copies those items to the stack. The items are placed in a list if \texttt{ALL} was chosen.</td>
</tr>
<tr>
<td>UP</td>
<td>Moves up one level in the menu structure, returning to the Main menu.</td>
</tr>
<tr>
<td>----</td>
<td>---------------------------------------------------------------------</td>
</tr>
<tr>
<td>VIEW</td>
<td>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.</td>
</tr>
<tr>
<td>EXIT</td>
<td>Quits the Spice48 Pac to the HP 48SX stack.</td>
</tr>
<tr>
<td>ENTER</td>
<td>Moves down one level in the menu structure, entering the analysis parameter screen.</td>
</tr>
<tr>
<td>ON - PREV</td>
<td>Dumps the current screen to an IR printer.</td>
</tr>
</tbody>
</table>

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 **ATN** to clear the command line (if necessary) and then press **ENTER** or **ATN** 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 **MICTR** to enter the value .000003). More exponent softkeys can be accessed by pressing **MK1**.
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 QUIT 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 QUIT 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

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analysis Parameter</td>
<td><strong>MAIN</strong> <strong>-STK</strong> <strong>PRINT</strong> <strong>CALC</strong> <strong>FONT</strong> <strong>UP</strong></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>CALC</strong></td>
<td>Begins an analysis, using the specified analysis parameters.</td>
</tr>
<tr>
<td><strong>FONT</strong></td>
<td>Toggles between the small and large fonts.</td>
</tr>
<tr>
<td><strong>MAIN</strong></td>
<td>Returns to the Main menu.</td>
</tr>
<tr>
<td><strong>PRINT</strong></td>
<td>Prompts for <strong>ONE</strong> or <strong>ALL</strong> to select items, and then sends those items to an IR printer.</td>
</tr>
<tr>
<td><strong>-STK</strong></td>
<td>Prompts for <strong>ONE</strong> or <strong>ALL</strong> to select items, and then copies those items to the stack. The items are placed in a list if <strong>ALL</strong> was chosen.</td>
</tr>
<tr>
<td><strong>UP</strong></td>
<td>Moves up one level in the menu structure, returning to the Analysis Manager.</td>
</tr>
<tr>
<td><strong>ATN</strong></td>
<td>Quits the Spice48 Pac to the HP 48SX stack. The element is saved automatically.</td>
</tr>
<tr>
<td><strong>ENTER</strong></td>
<td>Prompts for the value of the parameter currently selected by the pointer.</td>
</tr>
<tr>
<td><strong>ON-PREV</strong></td>
<td>Dumps the current screen to an IR printer.</td>
</tr>
<tr>
<td><strong>ON-NEW</strong></td>
<td>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.</td>
</tr>
</tbody>
</table>

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:

![Operating Point Analysis Screen]

You should now enter or edit the parameter values. When you have finished, press \texttt{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 \texttt{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 **ATN** 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 Parameter Screen]

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 **4** to quit Spice8.)

- **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 **4** to quit Spice8.)

- **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:

![Continue last TR calculation with advanced values?]

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 **EXIT** 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:

1. Press \([\text{LIBRARY}]\) to display all libraries available to your HP 48SX.
2. Find and press \([\text{SPICE}]\) to enter the Spice48 Pac library directory.
3. Press the first softkey, \([\text{SPICE}]\), to start the Spice48 Pac.
4. At the Main menu, make sure the pointer is at "Node Manager" and press \([\text{ENTER}]\).

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."): 
If the current circuit has no nodes, the screen will display, "Press ADD."

Summary of Operations

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node Manager</td>
<td>ADD DEL EDIT REFER UP</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>ADD</td>
<td>Adds one or more new nodes to the current circuit.</td>
</tr>
<tr>
<td>DEL</td>
<td>Deletes the node currently selected by the cursor.</td>
</tr>
<tr>
<td>EDIT</td>
<td>Edits the node currently selected by the cursor.</td>
</tr>
<tr>
<td>REFER</td>
<td>Sets the node currently selected by the cursor to be the reference node for the current circuit.</td>
</tr>
<tr>
<td>UP</td>
<td>Moves up one level in the menu structure, returning to the Main menu.</td>
</tr>
<tr>
<td>QUIT</td>
<td>Quits the Spice48 Pac to the HP 48SX stack.</td>
</tr>
<tr>
<td>ENTER</td>
<td>Edits the node currently selected by the cursor, or if no nodes exist, adds one or more new nodes to the current circuit</td>
</tr>
<tr>
<td>DUMP</td>
<td>Dumps the current screen to an IR printer.</td>
</tr>
</tbody>
</table>

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:

```
FRG HOME SPARCOM SPICE4BD BJTDC
Enter node name(s):
+ nodel node2 ...
(names)
```

Type in as many new node names as you wish (separated by spaces) and press \[ENTER\], or to abort the add operation, press \[ATN\] to clear the command line (if necessary), and then press \[ENTER\] or \[ATN\] 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 \[ATN\] 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 \[ATN\] to clear the command line (if necessary) and then press \[ENTER\] or \[ATN\] 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:

1. Press \[ \text{ [ ] } \] 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 “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.

<table>
<thead>
<tr>
<th>Item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set Node Coordinates</td>
<td>Edits node coordinates.</td>
</tr>
<tr>
<td>Set Elmt Coordinates</td>
<td>Edits element coordinates or automatically generation.</td>
</tr>
<tr>
<td>Set Wire Coordinates</td>
<td>Adds, deletes, and edits wire coordinates or automatically generates them.</td>
</tr>
<tr>
<td>View Schematic</td>
<td>Displays the schematic.</td>
</tr>
<tr>
<td>Print Schematic</td>
<td>Prints the schematic.</td>
</tr>
<tr>
<td>Clear All Graphics</td>
<td>Deletes all coordinates and the schematic.</td>
</tr>
</tbody>
</table>

**Summary of Operations**

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Graphics Manager</td>
<td><strong>MAIN</strong></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>FONT</strong></td>
<td>Toggles between the small and large fonts.</td>
</tr>
<tr>
<td><strong>MAIN</strong></td>
<td>Returns to the Main menu.</td>
</tr>
<tr>
<td><strong>PRINT</strong></td>
<td>Prompts for <strong>ONE</strong> or <strong>ALL</strong> to select items, and then sends those items to an IR printer.</td>
</tr>
<tr>
<td><strong>-STK</strong></td>
<td>Prompts for <strong>ONE</strong> or <strong>ALL</strong> to select items, and then copies those items to the stack. The items are placed in a list if <strong>ALL</strong> was chosen.</td>
</tr>
<tr>
<td><strong>UP</strong></td>
<td>Moves up one level in the menu structure, returning to the Main menu.</td>
</tr>
</tbody>
</table>
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 ESC to clear the command line (if necessary) and then press ENTER or ESC 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."): 

<table>
<thead>
<tr>
<th>Nodes:</th>
</tr>
</thead>
<tbody>
<tr>
<td>(0.2) SQLBDC (0.0) GAIN (0.2) INPUT</td>
</tr>
<tr>
<td>(1.2) BASE (0.3) EMIT (2.2) EMIT</td>
</tr>
<tr>
<td>(3.2) AMME (4.2) COLLE (5.2) OUTPUT</td>
</tr>
</tbody>
</table>

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:

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 ALT to clear the command line (if necessary), and then press ENTER or ALT 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 ALT to quit Spice48.

Summary of Operations

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node Coordinates</td>
<td>MAIN</td>
</tr>
</tbody>
</table>
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.”):

<table>
<thead>
<tr>
<th>Elements:</th>
</tr>
</thead>
<tbody>
<tr>
<td>(0.1) V5</td>
</tr>
<tr>
<td>(1.2) R1</td>
</tr>
<tr>
<td>(2.1) E1</td>
</tr>
<tr>
<td>(4.2) M1</td>
</tr>
<tr>
<td>(4.5) C3</td>
</tr>
</tbody>
</table>

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:
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 EXIT to clear the command line (if necessary), and then press ENTER or EXIT 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 EXIT to quit Spice48.

Summary of Operations

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element Coordinates</td>
<td>MAIN EDIT AUTO CLEAR UP</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>AUTO</td>
<td>Automatically generates element coordinates and orientations from the node coordinates.</td>
</tr>
<tr>
<td>CLEAR</td>
<td>Clears all of the current element coordinates and orientations.</td>
</tr>
</tbody>
</table>
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.”):

<table>
<thead>
<tr>
<th>Wires:</th>
</tr>
</thead>
<tbody>
<tr>
<td>(0.0) (5.0) (0.0) (1.0) (0.0) (5.0)</td>
</tr>
<tr>
<td>(0.0) (2.0) (0.0) (2.0) (1.3) (5.3)</td>
</tr>
<tr>
<td>(1.1) (1.0) (2.1) (3.1) (4.3) (6.3)</td>
</tr>
<tr>
<td>(5.1) (5.0) (6.2) (6.0)</td>
</tr>
</tbody>
</table>

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 END or EDIT. For adding and editing, you will be prompted to enter or edit the coordinates of the selected wire.
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 **ATN** to clear the command line (if necessary), and then press **ENTER** or **ATN** 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 **ATN** to quit Spice48.

### Summary of Operations

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wire Coordinates</td>
<td><strong>ADD</strong> <strong>DEL</strong> <strong>EDIT</strong> <strong>AUTO</strong> <strong>CLEAR</strong> <strong>UP</strong></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>ADD</strong></td>
<td>Prompts to enter the coordinates of a new wire and adds it.</td>
</tr>
<tr>
<td><strong>AUTO</strong></td>
<td>Automatically generates wire coordinates from the node coordinates and element coordinates and orientations.</td>
</tr>
<tr>
<td><strong>CLEAR</strong></td>
<td>Clears all of the current wire coordinates.</td>
</tr>
<tr>
<td><strong>DEL</strong></td>
<td>Deletes the wire currently selected by the cursor.</td>
</tr>
<tr>
<td>EDIT</td>
<td>Prompts to edit the coordinates of the wire currently selected by the cursor.</td>
</tr>
<tr>
<td>------</td>
<td>--------------------------------------------------------------------------</td>
</tr>
<tr>
<td>UP</td>
<td>Moves up one level in the menu structure, returning to the Graphics Manager.</td>
</tr>
<tr>
<td>EXIT</td>
<td>Quits the Spice48 Pac to the HP 48SX stack.</td>
</tr>
<tr>
<td>ENTER</td>
<td>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.</td>
</tr>
<tr>
<td>QUIT</td>
<td>Dumps the current screen to an IR printer.</td>
</tr>
</tbody>
</table>

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.”):

![BJTDC Schematic](image)

When you have finished viewing the schematic, press **ENTER** or **QUIT** 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:

Ready 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 [EXIT] 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:

Update schematic first?

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 [EXIT] 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:

1. Press \( \text{DispLib} \) to display all libraries available to your HP 48SX.
2. Find and press \( \text{SPICE} \) to enter the Spice48 Pac library directory.
3. Press the first softkey, \( \text{SPICE} \), to start the Spice48 Pac.
4. At the Main menu, make sure the pointer is at "Data File Manager" and press \( \text{Ent} \).

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 Spiccd8 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**

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Data File Manager</td>
<td>MAIN DEL XFER FONT UP</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>DEL</td>
<td>Deletes the data file currently selected by the pointer.</td>
</tr>
<tr>
<td>FONT</td>
<td>Toggles between the small and large fonts.</td>
</tr>
<tr>
<td>MAIN</td>
<td>Returns to the Main menu.</td>
</tr>
<tr>
<td>XFER</td>
<td>Transfers a data file to or from a computer.</td>
</tr>
<tr>
<td>UP</td>
<td>Moves up one level in the menu structure, returning to the Main menu.</td>
</tr>
<tr>
<td>EXIT</td>
<td>Quits the Spiccd8 Pac to the HP 48SX stack.</td>
</tr>
<tr>
<td>EXIT</td>
<td>Loads the data file currently selected by the pointer for viewing in the Output Manager.</td>
</tr>
<tr>
<td>DISPLAY</td>
<td>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.</td>
</tr>
<tr>
<td>DUMP</td>
<td>Dumps the current screen to an IR printer.</td>
</tr>
</tbody>
</table>

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 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 [Delete]. You will be prompted to verify the delete operation by pressing [Yes] to confirm or [No] or [Esc] 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].

Transfer to or from PC?

TO

FROM

At this prompt, press [TO]:

Place PC in Kermit server mode in desired directory...

OK

ABORT

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 do-
main 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 [Esc]. 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:

```
< HOME SPARCOM SPICE4BD FILES >
Enter file name:
+ name

<SKIP SKIP> <DEL> DEL> INS > INS
```

Press ENTER when you have finished typing the name:

```
Place PC in Kermit server mode in desired directory...

OK ABORT
```

Now, you must prepare your computer to be a Kermit server. For a PC, this in-
volves 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 [Esc]. 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:

1. Press \( \text{F1:CAT} \) to display all libraries available to your HP 48SX.

2. Find and press \text{SPICE} to enter the Spice48 Pac library directory.

3. Press the first softkey, \text{SPICE}, to start the Spice48 Pac.

4. The Output Manager cannot be accessed without data to view. To get data, you must do one of the following:

   a. Perform an analysis of the current circuit. To do this, make sure the pointer is at “Analysis Manager” and press \text{ENT}R. Then, move the pointer to the desired analysis type and press \text{ENT}R. Then, specify the analysis parameters and press \text{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):

\[
\begin{array}{l|c|c}
\text{freq, src, time} & \text{Circuit Nodes} & \text{Extra Nodes} \\
\downarrow & V_1 & I_1 \\
\downarrow & \downarrow & \downarrow \\
\downarrow & V_N & I_N \\
\end{array}
\]

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

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:

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:
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:

When you have finished, press ESC to accept the name and save the data slice, or to abort saving the data slice, press ESC to clear the command line (if necessary) and then press ESC or ESC 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 EXIT 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

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Data View</td>
<td>MAIN</td>
</tr>
<tr>
<td></td>
<td>SAVE</td>
</tr>
<tr>
<td></td>
<td>PLOT</td>
</tr>
<tr>
<td></td>
<td>DISP</td>
</tr>
<tr>
<td></td>
<td>PREC</td>
</tr>
<tr>
<td></td>
<td>UP</td>
</tr>
<tr>
<td></td>
<td>MAIN</td>
</tr>
<tr>
<td></td>
<td>STK</td>
</tr>
<tr>
<td></td>
<td>PRINT</td>
</tr>
<tr>
<td></td>
<td>MEM</td>
</tr>
<tr>
<td></td>
<td>FONT</td>
</tr>
<tr>
<td></td>
<td>UP</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>DISP</strong></td>
<td>Prompts for the display type.</td>
</tr>
<tr>
<td><strong>FONT</strong></td>
<td>Toggles between the small and large fonts.</td>
</tr>
<tr>
<td><strong>MAIN</strong></td>
<td>Returns to the Main menu.</td>
</tr>
<tr>
<td><strong>MEM</strong></td>
<td>Displays the size of the displayed data slice and the available free memory</td>
</tr>
<tr>
<td><strong>PLOT</strong></td>
<td>Plots the displayed data slice in a node plot or value plot.</td>
</tr>
<tr>
<td><strong>PREC</strong></td>
<td>Prompts for the display precision.</td>
</tr>
<tr>
<td><strong>PRINT</strong></td>
<td>Prompts for <strong>ONE</strong> or <strong>ALL</strong> to select items, and then sends those items to an IR printer.</td>
</tr>
<tr>
<td><strong>SAVE</strong></td>
<td>Saves the displayed data slice.</td>
</tr>
<tr>
<td><strong>STK</strong></td>
<td>Prompts for <strong>ONE</strong> or <strong>ALL</strong> to select items, and then copies those items to the stack. The items are placed in a list if <strong>ALL</strong> was chosen.</td>
</tr>
<tr>
<td><strong>UP</strong></td>
<td>Moves up one level in the menu structure, returning to the previous view or manager.</td>
</tr>
<tr>
<td><strong>EXIT</strong></td>
<td>Quits the Spice48 Pac to the HP 48SX stack.</td>
</tr>
<tr>
<td><strong>ENTER</strong></td>
<td>Displays the title, the label, and the value, all expanded to one screen.</td>
</tr>
<tr>
<td><strong>=</strong> - <strong>vist</strong></td>
<td>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.</td>
</tr>
</tbody>
</table>

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."):

![Node View Table]

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 un-marked 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 **ATN** to clear the command line (if necessary) and then press **ENTER** or **ATN** 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 \( \text{UP} \) to return to the previous manager, MAIN to return to the Main menu, or \( \text{EXIT} \) to quit Spiced.

If you accessed the Node View as a result of loading a data file, pressing \( \text{UP} \) 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 \( \text{UP} \) will return to the Analysis Manager.

Summary of Operations

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node View</td>
<td>MAIN, MARK, PLOT, NODE, ????, UP</td>
</tr>
<tr>
<td></td>
<td>MAIN, SAVE, MEM, DISP, PREC, UP</td>
</tr>
</tbody>
</table>

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

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."):

<table>
<thead>
<tr>
<th>DC: VBE Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>VBE</td>
</tr>
<tr>
<td>6.25</td>
</tr>
<tr>
<td>6.50</td>
</tr>
<tr>
<td>7.25</td>
</tr>
</tbody>
</table>

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 M Emin. 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 \[\text{Enter}\]. This will display the source value slice in the Data View. When you have finished viewing the source value slice, press \[\text{Up}\] 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 \[\text{Plot}\]. 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 \[\text{Plot}\]. 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 \[\text{Save}\]. You will be prompted to edit the default file name. When you have finished, press \[\text{Enter}\] to accept the name and save the source value slice(s), or to abort saving the source value slice(s), press \[\text{Esc}\] to clear the command line (if necessary) and then press \[\text{Enter}\] or \[\text{Ctrl}\] 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 \textbf{UP} to return to the previous manager, \textbf{MAIN} to return to the Main menu, or \textbf{ATN} to quit Spice48.

If you accessed the Source Value View as a result of loading a data file, pressing \textbf{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 \textbf{UP} will return to the Analysis Manager.

Summary of Operations

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Source Value View</td>
<td>MAIN</td>
</tr>
<tr>
<td></td>
<td>MARK</td>
</tr>
<tr>
<td></td>
<td>PLOT</td>
</tr>
<tr>
<td></td>
<td>NODE</td>
</tr>
<tr>
<td></td>
<td>SRC</td>
</tr>
<tr>
<td></td>
<td>UP</td>
</tr>
<tr>
<td></td>
<td>MAIN</td>
</tr>
<tr>
<td></td>
<td>SAVE</td>
</tr>
<tr>
<td></td>
<td>MEM</td>
</tr>
<tr>
<td></td>
<td>DISP</td>
</tr>
<tr>
<td></td>
<td>PREC</td>
</tr>
<tr>
<td></td>
<td>UP</td>
</tr>
</tbody>
</table>

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

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 Table]

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 [DATA] to clear the command line (if necessary) and then press [ENTER] or [DATA] 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 [EXIT] 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**

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency View</td>
<td><strong>MAIN</strong> <strong>MARK</strong> <strong>PLOT</strong> <strong>NODE</strong> <strong>FREQ</strong> <strong>UP</strong></td>
</tr>
<tr>
<td><strong>MAIN</strong></td>
<td><strong>SAVE</strong> <strong>MEM</strong> <strong>DISP</strong> <strong>PREC</strong> <strong>UP</strong></td>
</tr>
</tbody>
</table>

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

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. "): 

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 [EXIT] to clear the command line (if necessary) and then press [Enter] or [EXIT] 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 [EXIT] to quit Spice48.
If you accessed the Time View as a result of loading a data file, pressing \textbf{UP} will return to the Data File Manager, or if you accessed the Time View as a result of performing a TR analysis, pressing \textbf{UP} will return to the Analysis Manager.

\textbf{Summary of Operations}

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time View</td>
<td>\textbf{MAIN} \textbf{MARK} \textbf{PLOT} \textbf{NODE} \textbf{TIME} \textbf{UP} \textbf{MAIN} \textbf{SAVE} \textbf{MEM} \textbf{DISP} \textbf{PREC} \textbf{UP}</td>
</tr>
</tbody>
</table>

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

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

9: Output Manager 139
**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 `ENT`, or to adjust the vertical range, use the editing softkeys or press `ATN` to clear the command line and enter a new vertical range. After editing the vertical range, press `ENT` to accept the new vertical range and create a node plot, or to abort the node plot, press `ATN` to clear the command line (if necessary) and then press `ENT` or `ATN` 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."):

![Node Plot Example](image)

When you have finished viewing the node plot, press `EXIT` to return to the previous view.
Summary of Operations

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node Plot</td>
<td>None</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>⌈ ⌉</td>
<td>Scrolls node plots wider than the screen to view the extra nodes and/or the symbol key. When prefixed with ⌈ f ⌉, jumps to the edge of the node plot in the indicated direction.</td>
</tr>
<tr>
<td>⌈ ← ⌉</td>
<td>Exits the node plot and returns to the previous view.</td>
</tr>
<tr>
<td>⌈ → ⌉</td>
<td>Toggles between scrolling mode and Graphics Environment. (Graphics Environment is not useful for node plots.)</td>
</tr>
<tr>
<td>⌈ STO ⌉</td>
<td>If Graphics Environment is active, copies value plot (PICT) to the HP 48SX stack as a graphics object (GROB).</td>
</tr>
<tr>
<td>⌈ ON - PREV ⌉</td>
<td>Dumps the current screen to an IR printer.</td>
</tr>
</tbody>
</table>

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:

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 ATM 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 range, press **ENT**, or to adjust the range, use the editing softkeys or press **DEL** to clear the command line and enter a new range. After editing the range, press **ENT** to accept the new range, or to abort the value plot, press **Z** to clear the command line (if necessary) and then press **ENT** or **Z** 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."): 

142 9: Output Manager
When you have finished viewing the point plot, press \texttt{EXIT} 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."):

![Line Plot Example](image)

When you have finished viewing the line plot, press \texttt{EXIT} 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 \( \text{[MODE]} \) to toggle between the default view (scrolling mode) and the Graphics Environment. Useful functions in the Graphics Environment include SLOPE, COORD, F1, and F2. (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 \( \text{[ON]} \) to return to the previous view.

Summary of Operations

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value Plot</td>
<td>None</td>
</tr>
<tr>
<td>Graphics Environment</td>
<td>ZOOM Z-BOX CENT COORD LABEL FCN</td>
</tr>
<tr>
<td></td>
<td>DOT+ DOT- LINE TLINE BOX F1RCL</td>
</tr>
<tr>
<td></td>
<td>MARK REPL SUB DEL + _ KEYS</td>
</tr>
<tr>
<td>Function Analysis</td>
<td>ROOT ISECT SLOPE AREA EXTR EXIT</td>
</tr>
<tr>
<td></td>
<td>F X F' N axeQ</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>COORD</td>
<td>Displays the coordinates of the graphics cursor position.</td>
</tr>
<tr>
<td>F X</td>
<td>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.)</td>
</tr>
<tr>
<td>F'</td>
<td>Plots the first derivative the current function. (Equation plots only.)</td>
</tr>
<tr>
<td>FCN</td>
<td>Displays the Function Analysis menu for further analysis. (Function Analysis menu is useful only for equation plots.)</td>
</tr>
<tr>
<td>N axeQ</td>
<td>Switches to the next function in the value plot. (Equation plots only )</td>
</tr>
</tbody>
</table>
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.

Copies the rectangle bounded by the mark and the graphics cursor location to the HP 48SX stack as a graphics object (GROB).

Moves the graphics cursor in the indicated direction. When prefixed with \[J, moves the graphics cursor to the edge of the screen in the indicated direction.

Scrolls value plots wider than the screen to view the symbol key. When prefixed with \[L, 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 DTSP at any Output Manager view. You will be prompted for the display type:
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 [ON] 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 |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 Q**: 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](image)

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 [ON] 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 [FIN] to clear the command line (if necessary) and then press [FIN] or [FIN] 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 \[ &1[&9\] 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.

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

Summary of Operations

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Miscellaneous Info</td>
<td>MAIN -STK PRINT VIEW FONT UP</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>FONT</td>
<td>Toggles between the small and large fonts.</td>
</tr>
<tr>
<td>MAIN</td>
<td>Returns to the Main menu.</td>
</tr>
<tr>
<td>PRINT</td>
<td>Prompts for ONE or ALL to select items, and then sends those items to an IR printer.</td>
</tr>
<tr>
<td>-STK</td>
<td>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.</td>
</tr>
<tr>
<td>UP</td>
<td>Moves up one level in the menu structure, returning to the Main menu.</td>
</tr>
<tr>
<td>VIEW</td>
<td>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.</td>
</tr>
</tbody>
</table>
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):

<table>
<thead>
<tr>
<th>BLACK</th>
<th>BROWN</th>
<th>RED</th>
</tr>
</thead>
<tbody>
<tr>
<td>ORANGE</td>
<td>YELLOW</td>
<td>GREEN</td>
</tr>
<tr>
<td>BLUE</td>
<td>VIOLET</td>
<td>GREY</td>
</tr>
<tr>
<td>WHITE</td>
<td>SILVER</td>
<td>GOLD</td>
</tr>
</tbody>
</table>

You should now specify colors for the various bands of the resistor. To do this, move the cursor to the desired color and press 1, 2, 3, or 4, according to which band is of the selected color. When you have specified all of the bands, press CALC or ENTER to calculate the resistor value and tolerance. The results will be displayed on a separate screen.

At any point, press UP to return to Miscellaneous Info, MAIN to return to the Main menu, or EXIT to quit Spice48.

**Resistor Band Descriptions and Restrictions**

You can display a reference chart indicating the color restrictions for each band by pressing PICT. 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 ±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

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Resistor Colors</td>
<td>MAIN 4 5</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Sets 1st band to the color currently selected by the cursor.</td>
</tr>
<tr>
<td>2</td>
<td>Sets 2nd band to the color currently selected by the cursor.</td>
</tr>
<tr>
<td>3</td>
<td>Sets 3rd band to the color currently selected by the cursor.</td>
</tr>
<tr>
<td>4</td>
<td>Sets 4th band to the color currently selected by the cursor.</td>
</tr>
<tr>
<td>5</td>
<td>Sets 5th band to the color currently selected by the cursor.</td>
</tr>
<tr>
<td>CALC</td>
<td>Calculates the resistor value and tolerance from the colors.</td>
</tr>
<tr>
<td>FONT</td>
<td>Toggles between the small and large fonts.</td>
</tr>
<tr>
<td>PICT</td>
<td>Displays a reference chart indicating the color restrictions.</td>
</tr>
<tr>
<td>PRINT</td>
<td>Prompts for <strong>ONE</strong> or <strong>ALL</strong> to select items, and then sends those items to an IR printer.</td>
</tr>
<tr>
<td>STK</td>
<td>Prompts for <strong>ONE</strong> or <strong>ALL</strong> to select items, and then copies those items to the stack. The items are placed in a list if <strong>ALL</strong> was chosen.</td>
</tr>
<tr>
<td>UP</td>
<td>Moves up one level in the menu structure, returning to Miscellaneous Info.</td>
</tr>
<tr>
<td>VIEW</td>
<td>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.</td>
</tr>
<tr>
<td>QUIT</td>
<td>Quits the Spice48 Pac to the HP 48SX stack.</td>
</tr>
</tbody>
</table>
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.

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](image)

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 EXIT to quit Spice48.

**Summary of Operations**

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant Library</td>
<td>MAIN →STK PRINT UNITS FONT UP</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>FONT</td>
<td>Toggles between the small and large fonts.</td>
</tr>
<tr>
<td>MAIN</td>
<td>Returns to the Main menu.</td>
</tr>
<tr>
<td>PRINT</td>
<td>Prompts for ONE or ALL to select items, and then sends those items to an IR printer.</td>
</tr>
</tbody>
</table>
**SRS Prompts**

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.

**EXIT**

Quits the Spice48 Pac to the HP 48SX stack.

**ENTER**

Displays the screen title, the constant label, and the constant value, all expanded to one screen.

**ON-PREV**

Dumps the current screen to an IR printer.

**PRI**

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 **ENTER** to display the property description and value on a full screen, press **SRS** 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 **EXIT** 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:

```
HOME SPARCOM SPICE48
Enter component value:
→ real(1E-15-1E12)
```

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 ATN to clear the command line (if necessary) and then press ENTER or ATN 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:

```
<table>
<thead>
<tr>
<th>Pick tolerance:</th>
</tr>
</thead>
<tbody>
<tr>
<td>±5%</td>
</tr>
<tr>
<td>±10%</td>
</tr>
<tr>
<td>±15%</td>
</tr>
</tbody>
</table>
```

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 ATN 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 FSTK 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 ATN 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:

1 E-Mail

From Internet: support@sparcom.com
From Compuserve: >Internet:support@sparcom.com
From FidoNet: To:support@sparcom.com

2 Standard Mail

Sparcom Corporation
897 NW Grant Avenue
Corvallis, OR 97330
Attn: Technical Support Department

3 Telephone

(503) 757-8416
9 a.m. – 5 p.m. Pacific Standard Time

4 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:

1. Include your return address and a description of the problem.

2. Include a proof of purchase date if the warranty has not expired.

3. Include a purchase order, along with a check or credit card number and expiration date (VISA or MasterCard), to cover the standard repair charge.

4. 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:

1. Operating Temperature: 0 to 45°C (32 to 113°F).

2. Storage Temperature: –20 to 60°C (–4 to 140°F).

3. Operating and Storage Humidity: 90% relative humidity at 40°C (104°F) maximum.
# Appendix B

## Summary of Operations

### Summary of Screen Softkeys

<table>
<thead>
<tr>
<th>Screen</th>
<th>Softkeys</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analysis Manager</td>
<td>MAIN  FSTK  PRINT  VIEW  FONT  UP</td>
</tr>
<tr>
<td>Analysis Parameter</td>
<td>MAIN  FSTK  PRINT  CALC  FONT  UP</td>
</tr>
<tr>
<td>Circuit Manager</td>
<td>ADD  DEL  PACK  XFER  FONT  UP</td>
</tr>
<tr>
<td>Constant Library</td>
<td>MAIN  FSTK  PRINT  UNITS  FONT  UP</td>
</tr>
<tr>
<td>Data File Manager</td>
<td>MAIN  DEL  XFER  FONT  UP</td>
</tr>
<tr>
<td>Data View</td>
<td>MAIN  SAVE  PLOT  DISP  PREC  UP</td>
</tr>
<tr>
<td>Element Coordinates</td>
<td>MAIN  FSTK  PRINT  MEM  FONT  UP</td>
</tr>
<tr>
<td>Element Manager</td>
<td>ADD  DEL  EDIT  AUTO  CLEAR  UP</td>
</tr>
<tr>
<td>Element Parameter</td>
<td>MAIN  FSTK  PRINT  VIEW  FONT  UP</td>
</tr>
<tr>
<td>Frequency View</td>
<td>MAIN  MARK  PLOT  NODE  PREC  UP</td>
</tr>
<tr>
<td>Function Analysis</td>
<td>ROOT  ISECT  SLOPE  AREA  EXTR  EXIT</td>
</tr>
<tr>
<td>Graphics Environment</td>
<td>ZOOM  Z-BOX  CENT  COORD  LABEL  FCN  DOT  DOTT  LINE  TLINE  BOX  FIRE  REPL  SUB  DEL  +  -  KEYS</td>
</tr>
<tr>
<td>Graphics Manager</td>
<td>MAIN</td>
</tr>
<tr>
<td>------------------</td>
<td>-------</td>
</tr>
<tr>
<td>Main</td>
<td>ABOUT</td>
</tr>
<tr>
<td>Miscellaneous Info</td>
<td>MAIN</td>
</tr>
<tr>
<td>Node Coordinates</td>
<td>MAIN</td>
</tr>
<tr>
<td>Node Manager</td>
<td>ADD</td>
</tr>
<tr>
<td>Node Plot</td>
<td>None</td>
</tr>
<tr>
<td>Node View</td>
<td>MAIN</td>
</tr>
<tr>
<td></td>
<td>MAIN</td>
</tr>
<tr>
<td>Resistor Colors</td>
<td>MAIN</td>
</tr>
<tr>
<td></td>
<td>MAIN</td>
</tr>
<tr>
<td>Source Value View</td>
<td>MAIN</td>
</tr>
<tr>
<td></td>
<td>MAIN</td>
</tr>
<tr>
<td>Text Editing</td>
<td>SKIP</td>
</tr>
<tr>
<td>Time View</td>
<td>MAIN</td>
</tr>
<tr>
<td></td>
<td>MAIN</td>
</tr>
<tr>
<td>Transient Parameter</td>
<td>MAIN</td>
</tr>
<tr>
<td>Value Plot</td>
<td>None</td>
</tr>
<tr>
<td>Wire Coordinates</td>
<td>ADD</td>
</tr>
</tbody>
</table>

160  B: Summary of Operations
## Summary of Softkey Actions

<table>
<thead>
<tr>
<th>Key</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Sets 1st band to the color currently selected by the cursor.</td>
</tr>
<tr>
<td>2</td>
<td>Sets 2nd band to the color currently selected by the cursor.</td>
</tr>
<tr>
<td>3</td>
<td>Sets 3rd band to the color currently selected by the cursor.</td>
</tr>
<tr>
<td>4</td>
<td>Sets 4th band to the color currently selected by the cursor.</td>
</tr>
<tr>
<td>5</td>
<td>Sets 5th band to the color currently selected by the cursor.</td>
</tr>
<tr>
<td>ABOUT</td>
<td>Displays a screen containing the revision number and product information about the Spice48 Pac.</td>
</tr>
<tr>
<td>ADD</td>
<td>Adds a new circuit, element, node, or wire.</td>
</tr>
<tr>
<td>AUTO</td>
<td>Automatically generates wire coordinates or element coordinates and orientations from the node coordinates.</td>
</tr>
<tr>
<td>CALC</td>
<td>Begins an analysis, using the specified analysis parameters, or calculates the resistor value and tolerance from the colors.</td>
</tr>
<tr>
<td>CLEAR</td>
<td>Clears all of the current node, element, or wire coordinates.</td>
</tr>
<tr>
<td>COORD</td>
<td>Displays the coordinates of the graphics cursor position.</td>
</tr>
<tr>
<td>DEL</td>
<td>Deletes the circuit, element, node, wire, or data file currently selected by the pointer.</td>
</tr>
<tr>
<td>DEL=</td>
<td>Deletes all characters in the current word between the cursor's current position and the first character of the next word.</td>
</tr>
<tr>
<td>-DEL</td>
<td>Deletes all characters in the current word prior to the cursor.</td>
</tr>
<tr>
<td>DISP</td>
<td>Prompts for the display type.</td>
</tr>
<tr>
<td>EDIT</td>
<td>Edits the element, node, or coordinates currently selected by the cursor.</td>
</tr>
<tr>
<td>F  X</td>
<td>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.)</td>
</tr>
<tr>
<td>F'</td>
<td>Plots the first derivative the current function. (Equation plots only.)</td>
</tr>
<tr>
<td>FCN</td>
<td>Displays the Function Analysis menu for further analysis. (Function Analysis menu is useful only for equation plots.)</td>
</tr>
<tr>
<td>FONT</td>
<td>Toggles between the small and large fonts.</td>
</tr>
<tr>
<td>Key</td>
<td>Description</td>
</tr>
<tr>
<td>------</td>
<td>-------------</td>
</tr>
<tr>
<td>FREC</td>
<td>Switches to the Frequency View.</td>
</tr>
<tr>
<td>INS</td>
<td>Toggles between insert and type-over modes.</td>
</tr>
<tr>
<td>MAIN</td>
<td>Returns to the Main menu.</td>
</tr>
<tr>
<td>MARK</td>
<td>Toggles output node, source value, frequency, or time currently selected by the cursor between marked and unmarked states.</td>
</tr>
<tr>
<td>MEM</td>
<td>Displays the size of the data slice(s) and the available free memory.</td>
</tr>
<tr>
<td>NODE</td>
<td>Switches to the Node View.</td>
</tr>
<tr>
<td>NXEO</td>
<td>Switches to the next function in the value plot. (Equation plots only.)</td>
</tr>
<tr>
<td>PACK</td>
<td>Packs the circuit currently selected by the pointer.</td>
</tr>
<tr>
<td>PICT</td>
<td>Displays a reference chart indicating the color restrictions.</td>
</tr>
<tr>
<td>PLOT</td>
<td>Plots one or more data slices in a node or value plot.</td>
</tr>
<tr>
<td>PREC</td>
<td>Prompts for the display precision.</td>
</tr>
<tr>
<td>PRNT</td>
<td>Prompts for \texttt{ONE} or \texttt{ALL} to select items, and then sends those items to an IR printer.</td>
</tr>
<tr>
<td>QUIT</td>
<td>Quits the Spice4 Pac to the HP 48SX stack.</td>
</tr>
<tr>
<td>REFER</td>
<td>Sets the node currently selected by the cursor to be the reference node for the current circuit.</td>
</tr>
<tr>
<td>SAVE</td>
<td>Saves one or more data slices.</td>
</tr>
<tr>
<td>SKIP</td>
<td>Moves the cursor to the beginning of the current word.</td>
</tr>
<tr>
<td>SKIP-</td>
<td>Moves the cursor to the beginning of the next word.</td>
</tr>
<tr>
<td>SLOPE</td>
<td>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.)</td>
</tr>
<tr>
<td>SRC</td>
<td>Switches to the Source Value View.</td>
</tr>
<tr>
<td>STK</td>
<td>Prompts for \texttt{ONE} or \texttt{ALL} to select items, and then copies those items to the stack. The items are placed in a list if \texttt{ALL} was chosen.</td>
</tr>
<tr>
<td>STK+</td>
<td>Activates the Interactive Stack, allowing arguments to be copied from the stack to the command line for editing by pressing \texttt{ECHO}.</td>
</tr>
<tr>
<td>SUB</td>
<td>Copies the rectangle bounded by the mark and the graphics cursor location to the HP 48SX stack as a graphics object (GROB).</td>
</tr>
</tbody>
</table>
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 [less than], jumps to the edge of the plot in the indicated direction.

(Graphics)
Moves the graphics cursor in the indicated direction. When prefixed with [less than], 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.

ENTER
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

A: Warranty and Service 165
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
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
circuit, 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

A: Warranty and Service 169
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