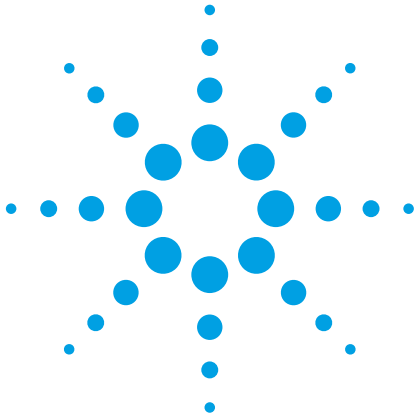
Agilent 3396 Series III
Agilent 3395 Integrator

Using Application Programs

Agilent Technologies

Agilent 3395/3396 Integrators



Manuals

These manuals may contain references to HP or Hewlett-Packard. Please note that Hewlett-Packard's former test and measurement, semiconductor products and chemicals analysis businesses are now part of Agilent Technologies. The HP 3395/3396 Integrator referred to throughout this document is now the Agilent 3395/3396 Integrator.

Agilent Technologies

HP 3396 Series III Integrator
HP 3395 Integrator

Using Application Programs

Manual Part No.
03395−90305

Edition 2, January 1995
Printed in USA

Printing History

The information contained in this document may be revised without notice.

Hewlett-Packard makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. Hewlett-Packard shall not be liable for errors contained herein or for incidental, or consequential damages in connection with the furnishing, performance, or use of this material.

No part of this document may be photocopied or reproduced, or translated to another program language without the prior written consent of Hewlett-Packard Company.

First Edition - May 1992
Printed In U.S.A.

Second Edition - Jan 1995
Printed In U.S.A.

Safety Information

The HP 3395 or HP 3396 Integrator meets the following IEC (International Electrotechnical Commission) classifications: Safety Class 1, Transient Overvoltage Category II, and Pollution Degree 2.

This unit has been designed and tested in accordance with recognized safety standards and designed for use indoors. If the instrument is used in a manner not specified by the manufacturer, the protection provided by the instrument may be impaired.

Whenever the safety protection of the HP 3395 or HP 3396 Integrator has been compromised, disconnect the unit from all power sources and secure the unit against unintended operation.

WARNING

A WARNING CALLS ATTENTION TO A CONDITION OR POSSIBLE SITUATION THAT COULD CAUSE YOU OR OTHERS INJURY.

CAUTION

A Caution calls attention to a condition or possible situation that could damage or destroy the product or your work.

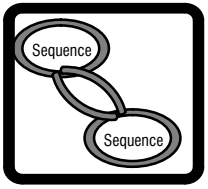
Important User Information for In Vitro Diagnostic Applications

This is a multipurpose product that may be used for qualitative or quantitative analyses in many applications. If used in conjunction with proven procedures (methodology) by a qualified operator, one of these applications may be In Vitro Diagnostic Procedures.

General instrument performance characteristics and instructions are included in this manual. Specific In Vitro Diagnostic procedures and methodology remain the choice and the responsibility of the user and are not included.

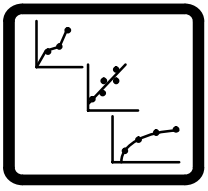
Contents

Chapter 6: Chaining Sequences



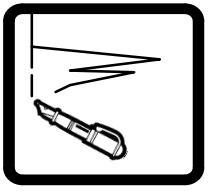
Chapter 6 shows that you can also automate runs by chaining together sequences. You can assign a different method to each sequence in the chain.

Chapter 7: Plotting a Calibration Curve



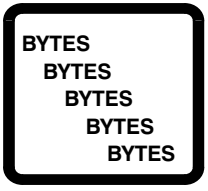
You can plot the response curve of a calibrated peak from a method or its calibration file. This chapter shows you how.

Chapter 8: Plotting a Baseline



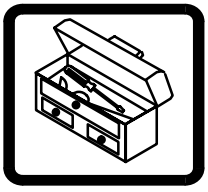
This chapter shows you how to run the Baseline program. You can run it interactively or schedule it as a postrun program. When this program runs, the chromatogram is replotted and its baseline is drawn.

Chapter 9: Maximizing Memory



The internal memory of the integrator is shared between data storage and space to run BASIC programs. This program makes space reserved for BASIC available for data storage.

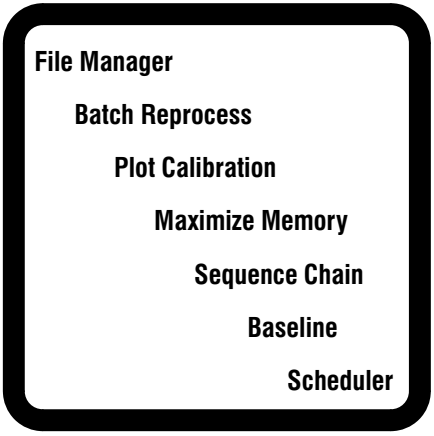
Appendix : Error Messages and Recovery Procedures



The Appendix lists error messages for the application programs and suggests possible causes and corrective user actions. It also contains error recovery procedures for autosequenced runs or sequences that have failed.

1

Introducing the Application Programs



In this chapter....

What Are Application Programs?

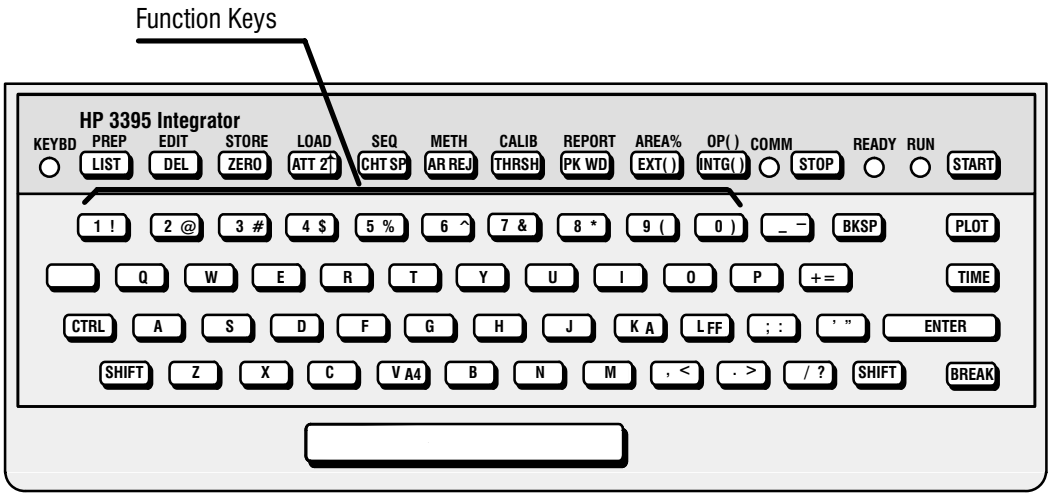
Your integrator contains a set of programs, called application programs, that extend your integrator's capabilities. Because they are permanently installed in an EPROM, you don't have to load an application program to run the integrator. In fact, you can run most of the programs simply by pressing the program's assigned function key.

Your integrator contains the following application programs:

Program	Application
Auto Start	Start autoscheduled runs or sequences
Autoscheduler	Schedule postrun programs
Autoname	Rename files automatically
File Manager	Copy, List, etc.
Batch Reprocess	Reprocess data files
Sequence Chaining	Automate analyses
Plot Calibration Curve	Check validity of multilevel calibrations
Plot Baseline	Check the baseline allocation
Maximize Memory	Toggles reserved memory space

Key Assignments for the Application Programs

The function keys are labelled 1 through 9, and 0. When you switch on the integrator, application programs are assigned to the function keys, as shown below the figure.



- 1 ! — File Manager
- 2 @ — Batch Reprocess
- 3 # — Plot Calibration Curve
- 4 ^ — Maximize Memory
- 6 ^ — Sequence Chaining
- 8 * — Plot Baseline
- 9 (— Autoscheduler
- 0) — Autostart

An Overview of the Application Programs

You can run most of the application programs by pressing the appropriate function key. Autoname must be autoscheduled as a postrun program. A short description of each application program follows.

The Autoscheduler

The Autoscheduler is used to specify an individual program or program chains to execute as postrun programs. It also provides the user interface for the autonaming program.

Auto Start

Use the **Auto Start** key to start an autoscheduled run or sequence.

Pressing the **Auto Start** key starts a program called `AUTO_TOP`. It performs a number of prerun tests to ensure that the postrun programs will execute.

Autoname (Automatic File Naming)

The Autoname program has no function key assignment. It can be scheduled to run as a postrun program using the Autoscheduler. It automatically renames the signal data file(s), processed peak file(s), and the report file(s).

File Manager

The File Manager program expands the file management capabilities of the HP 3395 Integrator. Commands include Copy, Delete, List, etc.

Batch Reprocess (Reprocessing Existing Data Files)

Through an interactive dialog, this program lets you specify a set of data files and calibration files to be reanalyzed. You may specify a method file, calibration file, or a sequence sample table to supply the sample information during reprocessing.

Sequence Chaining

Running this program starts a dialog, in which you can specify a chain of sequences and their corresponding methods, and select optional postrun programs. When you start the sequence chain, the first run of the sequence executes, followed by its postrun program (if any), followed by the next run and its postrun programs (if any), and so on until the sequence is finished.

Plot Calibration Curve

This program plots the selected peak's calibration curve based on the actual response factors and prints the curve's correlation coefficient.

Plot Baseline

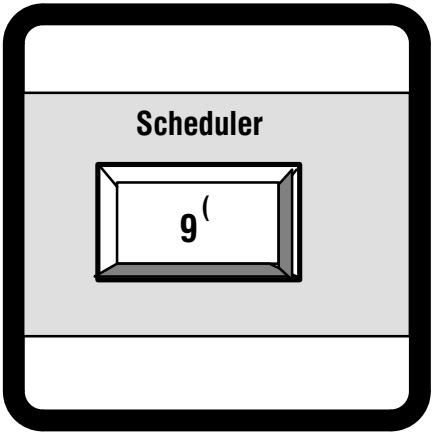
This program replots a chromatogram and draws in its baseline, using the signal file and the processed peak file from the original run. The signal and processed peak files supply the chromatographic and baseline information. The method determines the plot's appearance. You can plot baselines interactively from the keyboard or schedule baseline as a postrun program.

Maximize Memory

When the integrator is in default mode, 20 KBytes of memory space is specifically allocated to run BASIC programs. The space can be made available for data storage with this program.

2

Scheduling Postrun Programs



In this chapter...

Introducing the Autoscheduler

The primary function of the Autoscheduler is to schedule postrun programs that execute during the integrator's postrun phase. It also provides the user interface for the Autoname program and provides method and sequence assignments for Autoscheduler-controlled runs and sequences.

The Autoscheduler consists of three interrelated parts: the Autoscheduler dialog, the Autoscheduler file, and the Autoscheduler program.

Autoscheduler Dialog

You begin by interacting with the Autoscheduler dialog. Press function key [9], labelled **Scheduler**, and the dialog begins. The dialog prompts you to enter the names of the postrun programs you want to schedule. Your responses will be stored in the Autoscheduler file.

Autoscheduler File

When you have entered all the postrun programs, store the Autoscheduler file using the default name `M:AUTOCALL.UA1` and exit the dialog. You can store Autoscheduler files using other nondefault names; however the Autoscheduler program only operates with `M:AUTOCALL.UA1`.

Autoscheduler Program

When you start the autoscheduled run (or sequence), the Autoscheduler program executes the programs that you named in the default Autoscheduler file `M:AUTOCALL.UA1` *during the postrun portion of the run*.

Starting the Autoscheduler Dialog

When the integrator is first switched on, the application programs are assigned to their respective function keys. The assignment for the Autoscheduler is function key [9]. Pressing this key will start the Autoscheduler dialog.

1. Press function key [9] to start the Autoscheduler dialog.

```
      WELCOME TO THE HP 3396 POSTRUN AUTOSCHEDULER (REV C.00.01)


AT ANY ':' PROMPT:  'Q'[ENTER] QUITS
                    'S'[ENTER] STARTS OVER
IN EDIT MODE     :  'D'[ENTER] TERMINATES EDITING


*********************************************************************
NOTE: Default response to any [Y/N*] : prompt is negative.
      In EDIT MODE pressing only [ENTER] maintains the existing value
      of an option. In CREATE MODE pressing only [ENTER] sets the
      option to the default value, or to a space if no default exists.

*********************************************************************
```

The header information explains how to enter and exit the Autoscheduler dialog in both the create and edit modes.

Creating an Autoscheduler File

In an Autoscheduler file you can:

- Supply a data file name prefix for the Autoname program.
- Specify the method and sequence assignments for Autoscheduler-controlled runs and sequences.
- Schedule up to 20 postrun programs.

The following steps will illustrate how to create an Autoscheduler file by scheduling the Autoname program as a postrun program.

```
Edit an existing autocall scheduler file or create a new one
[E/C*]:
```

1. Press [ENTER] to create a new Autoscheduler file.

Your response to the first prompt determines whether you will create a new Autoscheduler file or edit one that already exists.

The default selection is **C**, indicated by the asterisk. It is entered automatically when you press **[ENTER]**.

```
RENAMING DATA FILES
-------------------
Data file name prefix for renaming: TESTA
```

2. Enter a prefix for Autoname result files.

This prompt is where you enter the file name prefix used by the Autoname program to rename result files. The prefix can have up to four characters plus an asterisk. Use of the asterisk is described in chapter 3, “Renaming Files Automatically.”

A valid prefix starts with a letter followed by any combination of letters, numbers, and the underscore character.

```
SPECIFYING METHOD AND SEQUENCE FILES
------------------------------------
Method file for autoscheduled run or sequence: M:METH2.MET
```

3. **Enter the method file to analyze the Autoscheduler-controlled run or sequence, or press [ENTER] to specify no method assignment.**

   The method specified here overrides a method specified in the sequence.

   If you do not supply the .MET file extension, the program does it for you.

   Note: If you press **[ENTER]** at the method prompt, no method is assigned and a warning message is printed.

```
WARNING - NO EXPLICIT METHOD SPECIFIED. THIS MAY RESULT
          IN UNUSABLE SYSTEM BEHAVIOR AT EXECUTION TIME.
```

   The analysis will be performed with the currently active method when you press **Auto Start**.

```
Sequence file for autoscheduled run or sequence: M:SEQ2.SEQ
```

4. **Enter the name of the sequence to automate runs, or press [ENTER] to specify no sequence assignment.**

   A method specified in the Autoscheduler dialog overrides a method specified in the sequence file.

   Note: If you press **[ENTER]** at the sequence prompt, no sequence is assigned and a warning message is printed.

```
WARNING - NO EXPLICIT SEQUENCE SPECIFIED. THIS MAY RESULT
          IN UNUSABLE SYSTEM BEHAVIOR AT EXECUTION TIME.
```

   The analysis will be performed with the currently active sequence when you press **Auto Start**.

```
SCHEDULING AUTOCALL PROGRAMS
----------------------------
Schedule postrun AUTOCALL programs [Y/N*]: Y
```

5. **Enter Y to schedule postrun programs.**

```
AUTOCALL filespec : E:AUTONAME.BAS
AUTOCALL filespec : [ENTER]
```

6. **Enter the name of the program you want to run as a postrun program. When there are no more program entries, press [ENTER] to continue on to the next prompt. If you do not specify a disk, it is defaulted to E (the EPROM).**

   If you do not supply the .BAS file extension, the program does it for you.

   The Autoname program is one of the application programs supplied with all HP 3395 and 3396 Integrators. Automatic file renaming prevents result files from being overwritten and lets you assign meaningful names to result files for future reference. See chapter 3 for more information about the Autoname program.

   You can schedule up to 20 postrun programs with the Autoscheduler.

```
Store the current Autoscheduler file ('N' re-edits) [Y/N*] Y
```

7. **Enter Y to store the entries you've made.**

```
Enter Autoscheduler file name [M:AUTOCALL.UA1*]: [ENTER]
```

8. **Press [ENTER] to store the current Autoscheduler file with its default name.**

   Valid file extensions are `.UA1`, `.UA2`, and `.UA3`. Some examples of valid Autoscheduler file names are `B:AUTOCALL.UA2` and `M:AUTOCALL.UA3`.

   - If you do not supply the .UA1 file extension, the program does it for you.
   - If you do not specify a disk drive, M will automatically be assigned.
   - If you do not specify a file name, `M:AUTOCALL.UA1` is automatically assigned as the Autoscheduler file.

When you create an Autoscheduler file, name it according to its function. For example, name a file `H20SAMP.UA1` that analyzes water samples.

To prepare the file for an autoscheduled run or sequence, rename the file to `M:AUTOCALL.UA1`, and, when the runs are finished, rename the file to its original name. This renaming operation is necessary because the Autoscheduler only works with files named `M:AUTOCALL.UA1`.

```
M:AUTOCALL.UA1 - Exists, OK to overwrite [Y/N*]: Y
```

9. The program prompts you with this message ONLY if an Autoscheduler file with the name you entered in step 8 already exists.

- Enter **Y** to overwrite the existing file with the new information.
- Enter **N** to keep the existing file. The dialog reprompts you for the name to store file.

```
Create another Autoscheduler file [Y/N*]: [ENTER]
```

10. Enter N or press [ENTER] to exit the dialog.

If you enter **Y**, the create dialog will restart.

Now you are ready to start the run. See "Starting the Automated Run or Sequence" on the next page.

Writing and Scheduling User Written Programs

The last statement in a user written BASIC program *must* chain back to the Autoscheduler program (`E:AUTO_SKD.BAS`). Users BASIC programs can be *developed* on the HP 3396; not an HP 3395 integrator.

Example:

```
10          PRINT "This is an example program"
20          PRINT ""
30          FOR D=1 to 22
40            PLOT 0,1312
50          NEXT D
60          END PLOT
70          PRINT ""
80          CHAIN "E:AUTO_SKD.BAS"
```

Starting the Automated Run or Sequence

After you have prepared the Autoscheduler file, you can start the run or the sequence.

You *must* use function key [0] to start an autoscheduled run or sequence. If you press the **[START]** button on the integrator, GC, etc., the run will start but the postrun programs will *not* execute.

After each run, the postrun program(s) specified in the `M:AUTOCALL.UA1` Autoscheduler file will execute. In this example, the postrun program is the Autoname program.

1. **Press function key [0] to initiate the START dialog.**

   When you press key [0], labelled **Auto Start**, a program called `AUTO_TOP` performs a number of prerun tests that check program space to ensure that the postrun programs will execute.

   ```
   VERIFYING FILES, BASIC WORKSPACE
   SETTING SEQ/MET, BASIC WORKSPACE
   AUTOSCHEDULER WORKFILE COMPATIBLE WITH SYSTEM
   ```

   When everything checks okay, you are prompted to start the run or sequence.

   ```
   Start a Run or Sequence or Quit [R*/S/Q]: [ENTER]
   ```

2. **Press [ENTER] to start the run, or enter S to start a sequence.**

If you need to abort the run at this point, enter **Q** to quit. Then reedit the method or sequence, or fix whatever needs fixing. When you are ready to start, press the **Auto Start** key.

The resulting run is shown on the next page.

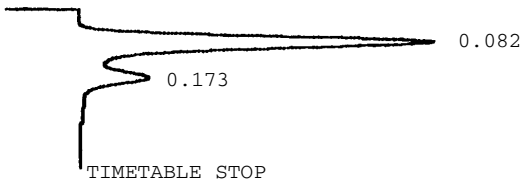
Example—Scheduling and Running the Autoname Program

The dialog and user responses below summarize the steps described in "Creating an Autoscheduler File."

```
Edit an existing autocall scheduler file or create a new one [E/C*]: C

RENAMING DATA FILES
-------------------
Data file name prefix for renaming: TEST

SPECIFYING METHOD AND SEQUENCE FILES
------------------------------------
Method file for autoscheduled run or sequence: M:METH2.MET
Sequence file for autoscheduled run or sequence: M:SEQ2.SEQ

SCHEDULING AUTOCALL PROGRAMS
----------------------------
Schedule postrun AUTOCALL programs [Y/N*]: Y
AUTOCALL filespec : E:AUTONAME.BAS
AUTOCALL filespec : [ENTER]
Store the current Autoscheduler file ('N' re-edits) [Y/N*] Y
Enter autoscheduler file name [M:AUTOCALL.UA1*]: [ENTER]
M:AUTOCALL.UA1 - Exists, ok to overwrite [Y/N*]: Y
Create another Autoscheduler file [Y/N*]: [ENTER]
```

Start the autoscheduled run by pressing the **Auto Start** key.

```
*  VERIFYING FILES, BASIC WORKSPACE

SETTING SEQ/MET, BASIC WORKSPACE

AUTOSCHEDULER WORKFILE COMPATIBLE WITH SYSTEM


Start a sequence or a run or Quit [R*/S/Q]: [ENTER]


* RUN #    1      AUG  28, 1994  13:46:51
START
```




```
 Closing signal file M:SIGNAL   .BNC
 Storing processed peaks to M:Q1EB7A13.PRO

 RUN#     1          AUG  28, 1994  13:46:51

 SIGNAL FILE:  M:SIGNAL.BNC
 PEAK FILE  :  M:Q1EB7A13.PRO
 AREA%
       RT      AREA  TYPE  WIDTH      AREA%
     .082     36970    B     .03     81.8568
     .173         2    V     9             9
              79010    V     .04     18.1431
 TOTAL AREA=448712     B     2             1
 MUL FACTOR=1.0000E+00
```

```
Signal file M:SIGNAL.BNC renamed M:TESTA001.BNC

Processed peak file M:Q1EC06E9.PRO renamed M:TESTA001.PRO


==============       END OF RUN       ==============
```

Editing an Autoscheduler File

In the edit mode, the dialog prints the prompt line, including the last user entry, followed by a colon edit prompt (:) on the line below.

An example of a prompt in the Autoscheduler edit mode is shown below:

```
Method file for realtime channel: M:METH2.MET

:
```

You can respond to the edit prompt in three ways:

- Press **[ENTER]** to preserve the original response.
- Type a new response and press **[ENTER]** to overwrite the original response.
- Press **[SPACE] [ENTER]** to delete the original response.

Note: The Autoscheduler file changes only when you store it. If you have partially edited an Autoscheduler file and make a mistake, you can quit or restart the editing session by typing **Q** or **S** respectively. You can enter a **D** at any time to specify that you are "done" editing the Autoscheduler file.

The following steps illustrate how to edit an existing Autoscheduler file to change the scheduled postrun programs.

```
Edit an existing autocall scheduler file or create a new one [E/C*]: E
```

1. Enter E to edit an existing Autoscheduler file.

```
AUTOCALL Scheduler file to edit [M:AUTOCALL.UA1*]: [ENTER]
```

2. **Press [ENTER] to edit the default Autoscheduler file, M:AUTOCALL.UA1, or enter the name of the file you want to edit.**

```
RENAMING DATA FILES
-------------------

Data file name prefix for renaming: TEST

: BH13
```

3. **If desired, type a new data file name prefix for renaming result files.**

```
SPECIFYING METHOD AND SEQUENCE FILES
------------------------------------

Method file for autoscheduled run or sequence: M:METH2.MET

: [ENTER]
```

4. **Press [ENTER] to leave the method specification unchanged, or enter a new method file name.**

```
Sequence for autoscheduled run or sequence: M:SEQ2.SEQ

: [ENTER]
```

5. **Press [ENTER] to leave the sequence specification unchanged, or enter a new sequence file name.**

```
SCHEDULING AUTOCALL PROGRAMS
----------------------------
Schedule postrun AUTOCALL programs [Y/N*]: Y

:[ENTER]
```

6. **Enter Y to schedule postrun programs.**

```
AUTOCALL filespec : E:AUTONAME.BAS

: [ENTER]
```

7. **Press [ENTER] to leave the postrun program specification unchanged.**

```
AUTOCALL filespec : E:BASELINE.BAS

: E:AUTO_2CH.BAS
```

8. Enter a new postrun program specification.

If you do not specify a disk, it is defaulted to E (the EPROM).

```
AUTOCALL filespec : E:USERPROG.BAS

: [SPACE] [ENTER]
```

9. Press [SPACE] [ENTER] to delete the postrun program specification.

When you delete a stored postrun program, all of the programs that follow it are deleted also. If you delete a postrun program from within a list, you will have to retype all of the postrun programs that were deleted along with the desired program deletion.

```
Store the current Autoscheduler file ('N' re-edits) [Y/N*]: Y
```

10. Enter Y to store the edited Autoscheduler file.

If you make a mistake when editing:

Type **Q** to quit the edit mode.

Type **S** to restart the edit mode.

Then you can then go back to editing, and the original Autoscheduler file will remain unchanged.

```
Enter autoscheduler file name [M:AUTOCALL.UA1*]: [ENTER]
```

11. Press [ENTER] to store the current Autoscheduler file with its default name, or enter a new name for the edited file.

Valid file extensions are .UA1, .UA2, and .UA3. Some examples of valid Autoscheduler file names are: `B:AUTOCALL.UA2` and `M:AUTOCALL.UA3`.

- If you do not supply the .UA1 file extension, the program does it for you.
- If you do not specify a disk drive, M: will automatically be assigned.

When you create an Autoscheduler file, name it according to its function. For example, name a file that analyzes water samples `M:H20SAMP.UA1`.

To prepare the file for an autoscheduled run or sequence, rename the file to `M:AUTOCALL.UA1`, and, when the runs are finished, rename the file to its original name. This renaming operation is necessary because the Autoscheduler only works with files named `M:AUTOCALL.UA1`.

```
M:AUTOCALL.UA1 - Exists, ok to overwrite [Y/N*]: Y
```

12. The program prompts you with this message if an Autoscheduler file with that name already exists.

- Enter **Y** to overwrite the existing file with the new information.
- Enter **N** to keep the existing file. The dialog will prompt you for the name to store file.

```
Create another Autoscheduler file [Y/N*]: [ENTER]
```

13. Enter N or press [ENTER] to exit the dialog.

If you enter **Y**, the create dialog will start.

Example—Editing an Autoscheduler File

The dialog and user responses below summarize the steps described in "Editing an Autoscheduler File."

```
Edit an existing autocall scheduler file or create a new one [E/C*]: E

AUTOCALL Scheduler file to edit [M:AUTOCALL.UA1*]: [ENTER]


RENAMING DATA FILES
-------------------
Data file name prefix for renaming: TEST
: BH13


SPECIFYING METHOD AND SEQUENCE FILES
------------------------------------
Method file for autoscheduled run or sequence: M:METH2.MET
: [ENTER]

Sequence for autoscheduled run or sequence: M:SEQ2.SEQ
: [ENTER]


SCHEDULING AUTOCALL PROGRAMS
----------------------------
Schedule postrun AUTOCALL programs [Y/N*]: Y
:[ENTER]

AUTOCALL filespec : E:AUTONAME.BAS
: [ENTER]AUTOCALL filespec : E:BASELINE.BAS

: E:AUTO_2CH.BAS

AUTOCALL filespec : E:USERPROG.BAS
: [SPACE] [ENTER]

Store the current Autoscheduler file ('N' re-edits) [Y/N*]: Y

Enter autoscheduler file name [M:AUTOCALL.UA1*]: [ENTER]

M:AUTOCALL.UA1 - Exists, OK to overwrite [Y/N*]: Y

Create another Autoscheduler file [Y/N*]: [ENTER]
```

- File name prefix TEST is replaced by BH13.
- The specified method and sequence files remain unchanged.
- Postrun programs AUTONAME and BASELINE remain unchanged while USERPROG is deleted.
- The new file is stored as AUTOCALL.UA1 overwriting the old file.

Creating and Storing Multiple Autoscheduler Files

You can create several different Autoscheduler files by storing each one with a unique name. Having several Autoscheduler files on hand is useful when:

- You frequently run different sets of samples that require different postrun programs. Rather than creating a new file each time, prepare several Autoscheduler files and store them, one for each type of analysis.
- You are chaining sequences. The sequence chaining program lets you specify a unique Autoscheduler file for any sequence in the chain. To schedule postrun programs in a sequence chain, you *must* prepare the Autoscheduler files first, and then start the sequence chain program.

The following example illustrates how to create and store an Autoscheduler file for future use.

You can use this procedure to create as many Autoscheduler files as you need. Remember to assign a unique name to each file, one that clearly indicates its function.

```
Edit an existing autocall scheduler file or create a new one [E/C*]: C
```

1. **Enter E to edit the default Autoscheduler file, or press [ENTER] to create a new file.**

```
RENAMING DATA FILES
-------------------
Data file name prefix for renaming: BH13
```

2. **Enter the Autoname prefix.**

```
SPECIFYING METHOD AND SEQUENCE FILES
------------------------------------
Method file for autoscheduled run or sequence: M:METH2.MET

Sequence for autoscheduled run or sequence: M:SEQ2.SEQ
```

3. **Enter the method and sequence files for the Autoscheduler-controlled run or sequence.**

```
SCHEDULING AUTOCALL PROGRAMS
 ----------------------------
Schedule postrun AUTOCALL programs [Y/N*]: Y

AUTOCALL filespec: E:BASELINE.BAS

AUTOCALL filespec: E:AUTONAME.BAS

AUTOCALL filespec: [ENTER]
```

4. **Enter the postrun program names. You can schedule up to 20 postrun programs.**

```
Store the current Autoscheduler file ('N' re-edits) [Y/N*] ? Y

Enter Autoscheduler file name [M:AUTOCALL.UA1*]: M:WATR-
CALB.UA1
```

5. **When the dialog prompts you to store the Autoscheduler file, enter a name that somehow indicates the type of analysis.**

   For example:

   Use `M:WATRCONT.UA1` to name an Autoscheduler file used to analyze contaminated water.

   Do not use the default name `M:AUTOCALL.UA1`.

```
Create another AUTOCALL scheduler file [Y/N*]: N
```

6. **Press [ENTER] to select N, and exit the dialog.**

Use this procedure to create as many Autoscheduler files as you need. Remember to assign a unique name that clearly indicates the analysis to each file.

Recalling a Stored Autoscheduler File

Assume that you have created and stored a number of Autoscheduler files under various names. Now, you want to recall one of them to run with a particular set of samples. You created this file several weeks ago and named it `M:H20SAMP.UA1` (for water-sample analysis).

To recall your Autoscheduler file for a run or sequence, rename it to `M:AUTOCALL.UA1`. The renaming operation is necessary because the Autoscheduler only works with files named `M:AUTOCALL.UA1`.

When the runs are finished, return the file to its original name, in this case `M:H20SAMP.UA1`.

Stopping an Autoscheduled Run or Sequence

Press [STOP].

Restarting an Autoscheduled Run or Sequence

There are two ways that the run or sequence can be stopped:

- Pressing [STOP]
- Fatal error detected by an application program.

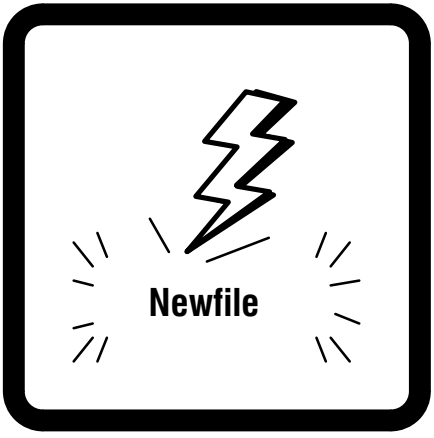
When an autoscheduled run or sequence is stopped with the [STOP] key, simply press key [0], the **Auto Start** key, to restart the run or sequence.

When an autoscheduled run or sequence is stopped prematurely due to a fatal error, an error message is printed.

1. Follow the instructions given by the message.
2. If the message contains no instructions, look up the possible cause and user action for the error message(s) or press key [0] to start error recovery.
3. Press key [0] again to start another run or sequence.

3

Renaming Files Automatically



In this chapter...

The Autoname Program

Autoname is an application program that automatically renames the signal data, processed peak, and report files after each run. There are two major reasons for renaming these files.

- To provide more descriptive file names for archiving.

  The default names for the signal data, processed peak, and report files are `SIGNAL.RAW` (or `SIGNAL.BNC`), `Qnnnnnnn.PRO`, and `Qnnnnnnn.RPT`, respectively, where `nnnnnnn` is a hexadecimal number based on the integrator calendar, time of day, and injection time. Sometimes this file name format is not descriptive enough.

- To prevent overwriting the signal data file.

  The processed peak and report files are assigned unique names after each run.

Note: Without Autoname, the signal data file, named `M:SIGNAL.RAW` or `M:SIGNAL.BNC`, is overwritten after each run unless it is stored on disk.

File Renaming Formats

The Autoname program can rename result files in two different formats:

- A user-specified prefix appended with the run number: `TEST002.BNC, TEST002.PRO, TEST002.RPT`

  Where: prefix = TEST, and run number = 002

- The sample names from the active sample table: `SLUDGE_1.BNC, SLUDGE_1.PRO, SLUDGE_1.RPT`

  Where: sample name for the run is SLUDGE_1.

Starting the Autoscheduler Dialog

The Autoscheduler program contains the dialog for scheduling the Autoname program as a postrun program.

When switched on, the Autoscheduler is assigned to function key [9]. Unless you've changed the key assignment, pressing this key will start the Autoscheduler dialog.

1. **Press function key [9] to start the Autoscheduler dialog.**

```
      WELCOME TO THE HP 3396 POSTRUN AUTOSCHEDULER (REV A.00.04)


AT ANY ':' PROMPT:  'Q'[ENTER] QUITS
                    'S'[ENTER] STARTS OVER
IN EDIT MODE     :  'D'[ENTER] TERMINATES EDITING


*********************************************************************
NOTE: Default response to any [Y/N*] : prompt is negative.
      In EDIT MODE pressing only [ENTER] maintains the existing value
      of an option. In CREATE MODE pressing only [ENTER] sets the
      option to the default value, or to a space if no default exists.
*********************************************************************
```

The header information explains how to enter and exit the Autoscheduler dialog in both the create and edit dialog.

```
Edit an existing Autoscheduler file or create a new one [E/C*]:
```

2. **Press [ENTER] to create a new Autoscheduler file.**

Your response to the first prompt determines whether you will create a new Autoscheduler file or edit one that already exists.

The default selection is **C**, indicated by the asterisk. It is entered automatically when you press **[ENTER]**.

3. **For the remaining steps, see the renaming mode you want to use:**

- Renaming Files with Sample Names (page 3-4)
- Renaming Files with a Prefix and Run Number (page 3-6)

Renaming Files

When using a sequence to run a series of samples, you can rename the result files of each run. The result files are the signal data file, processed peak file, and report file.

Result files can be renamed using three different formats:

- Sample names from the sequence sample table.
- Prefix with appended run number.
- Prefix with appended run number and vial number.

Renaming Files with Sample Names

To rename files using sample names, you must schedule Autoname as a postrun program *without* supplying a file name prefix.

```
RENAMING DATA FILES
-------------------
Data file name prefix for renaming: [ENTER]
```

3. **Press [ENTER] in response to the renaming prompt. This specifies that no prefix will be used for renaming files.**

```
SPECIFYING METHOD AND SEQUENCE FILES
------------------------------------
Method file for autoscheduled run or sequence: M:METH2.MET
```

4. **Enter the method file to analyze the Autoscheduler-controlled run or sequence, or press [ENTER] to specify no method assignment.**

   If automate runs with a sequence, a method specified in the Autoscheduler dialog overrides a method specified in the sequence.

Note: If you press **[ENTER]** at the method prompt, no method is assigned and a warning message is printed.

```
WARNING – NO EXPLICIT METHOD SPECIFIED. THIS MAY RESULT
          IN UNUSABLE SYSTEM BEHAVIOR AT EXECUTION TIME.
```

The analysis will be performed with *whatever is the currently active method.*

```
Sequence file for autoscheduled run or sequence: M:SEQ2.SEQ
```

5. **Enter the name of the sequence to automate runs. To specify no sequence assignment, press [ENTER] .**

   A method specified in the Autoscheduler dialog overrides a method specified in the sequence file.

   Note: If you press **[ENTER]** at the sequence prompt, no sequence is assigned and a warning message is printed.

   ```
   WARNING – NO EXPLICIT SEQUENCE SPECIFIED. THIS MAY RESULT
             IN UNUSABLE SYSTEM BEHAVIOR AT EXECUTION TIME.
   ```

   The analysis will be performed with *whatever is the currently active sequence.*

How the Files Are Renamed

During a run, the file names for the run come from the sample table of the active sequence.

For example, when the sample name for the run is `SLUDGE_1`, the signal data, processed peak, and report files will be named `SLUDGE_1.BNC`, `SLUDGE_1.PRO`, `SLUDGE_1.RPT`, respectively.

6. **The next steps are "Scheduling the Autoname Program" and "Starting the Run or Sequence." Go to page 3-8.**

Renaming Files with a Prefix and Run Number

You can set up the Autoname program to rename the signal data file, processed peak file, and report file of each run with a user-specified prefix and appended run number.

To rename the run file this way, supply a file name prefix and schedule Autoname as a postrun program.

```
RENAMING DATA FILES
-------------------
Data file name prefix for renaming:TEST
```

3. Enter TEST as the prefix for the renamed files.

The prefix can be up to four characters long.

A three-digit counter corresponding to the run number is appended to this prefix to form the file names. As the counter advances with each run, unique file names are created.

```
SPECIFYING METHOD AND SEQUENCE FILES
------------------------------------
Method file for autoscheduled run or sequence: M:METH2.MET
```

4. Enter the method file to analyze the Autoscheduler-controlled run or sequence, or press [ENTER] to specify no method assignment.

If automate runs with a sequence, a method specified here in the Autoscheduler dialog overrides a method specified in the sequence.

Note: If you press **[ENTER]** at the method prompt, no method is assigned and a warning message is printed.

```
WARNING – NO EXPLICIT METHOD SPECIFIED. THIS MAY RESULT
          IN UNUSABLE SYSTEM BEHAVIOR AT EXECUTION TIME.
```

The analysis will be performed with *whatever is the currently active method*.

```
Sequence file for autoscheduled run or sequence: M:SEQ2.SEQ
```

5. **Enter the name of the sequence to automate runs, or press [ENTER] to specify no sequence assignment.**

   A method specified in the Autoscheduler dialog overrides a method specified in the sequence file.

   Note: If you press **[ENTER]** at the sequence prompt, no sequence is assigned and a warning message is printed.

   ```
   WARNING – NO EXPLICIT SEQUENCE SPECIFIED. THIS MAY RESULT
             IN UNUSABLE SYSTEM BEHAVIOR AT EXECUTION TIME.
   ```

   The analysis will be performed with *whatever is the currently active sequence.*

How the Files Are Renamed

During a run, the data file name prefix and the last run number are used to produce the signal data file, processed peak file, and report file names for the run.

For example, when the prefix is `TEST003` and run number is `003`, the signal data, processed peak, and report files will be named `TEST003.BNC`, `TEST003.PRO`, `TEST003.RPT`, respectively.

6. **The next steps are "Scheduling the Autoname Program" and "Starting the Run or Sequence." Go to page 3-8.**

Scheduling the Autoname Program

```
SCHEDULING AUTOCALL PROGRAMS
----------------------------
AUTOCALL filespec: E:AUTONAME.BAS

AUTOCALL filespec: [ENTER]
```

1. **Enter Autoname as a postrun program then press [ENTER] to continue on to the next prompt. If you do not specify a disk, it will be defaulted to E disk, the EPROM.**

   Automatic file renaming prevents result files from being overwritten and lets you assign meaningful names to result files for future reference.

   You can schedule up to 20 postrun programs with the Autoscheduler.

   Note: If you are also scheduling the Baseline program, Autoname must be scheduled as the first AUTOCALL filespec.

```
Store the current Autoscheduler file ('N' re-edits) [Y/N*] ? Y
```

2. **Enter Y to store the entries you've made.**

```
Enter Autoscheduler file name [M:AUTOCALL.UA1*]: [ENTER]
```

3. **Press [ENTER] to keep the current Autoscheduler file name.**

```
M:AUTOCALL.UA1 - Exists, ok to overwrite [Y/N*]: Y
```

4. **The program prompts you with this message if an Autoscheduler file with that name already exists.**

   - Enter **Y** to overwrite the existing file with the new information.
   - Enter **N** to keep the existing file. The dialog will prompt you for the name to store file.

```
Create another Autoscheduler file [Y/N*]: [ENTER]
```

5. **Press [ENTER] to exit the dialog.**

   Now you are ready to start the run or sequence.

Starting the Run or Sequence

After you have scheduled the Autoname program, you can start the run or the sequence.

You *must* use function key [0] to start an autoscheduled run or sequence. If you press the **[START]** button on the integrator, GC, etc., the run will start but the postrun programs will *not* execute.

After each run, the postrun program(s) specified in the `M:AUTOCALL.UA1` Autoscheduler file will execute. In this example, the postrun program is the Autoname program.

1. **Press function key [0] to initiate the START dialog.**

   When you press key [0], labelled **Auto Start**, a program called `AUTO_TOP` performs a number of prerun tests that check program space to ensure that the postrun programs will execute.

```
*  VERIFYING FILES, BASIC WORKSPACE
SETTING SEQ/MET, BASIC WORKSPACE
AUTOSCHEDULER WORKFILE COMPATIBLE WITH SYSTEM
```

   When everything checks okay, you are prompted to start the run or sequence.

```
Start a sequence or a run or quit [R*/S/Q]: [ENTER]
```

2. **Press [ENTER] to start the run, or enter S to start a sequence.**

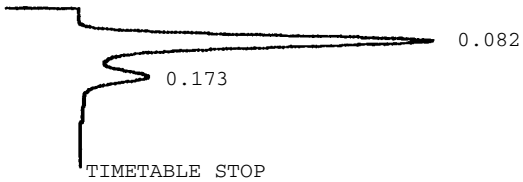
   If you need to abort the run at this point, enter **Q** to quit. When you are ready to start, press the **Auto Start** key.

   The dialog preparation and analytical run shown on the following pages illustrate renaming result files with a prefix and run number.

Example—Renaming Result Files with a Prefix and Run Number

```
Edit an existing autocall scheduler file or create a new one [E/C*]:

RENAMING DATA FILES
-------------------
Data file name prefix for renaming: TESTA [ENTER]


SPECIFYING METHOD AND SEQUENCE FILES
------------------------------------
Method file for autoscheduled run or sequence: M:METH2.MET

Sequence file for autoscheduled run or sequence: M:SEQ2.SEQ


RENAMING DATA FILES
-------------------
Data file name prefix for renaming:TEST


SPECIFYING METHOD AND SEQUENCE FILES
------------------------------------
Method file for autoscheduled run or sequence: M:METH2.MET

Sequence file for autoscheduled run or sequence: M:SEQ2.SEQ


SCHEDULING AUTOCALL PROGRAMS
----------------------------
AUTOCALL filespec: E:AUTONAME.BAS

AUTOCALL filespec: [ENTER]

Store the current Autoscheduler file ('N' re-edits) [Y/N*] ? Y

Enter Autoscheduler file name [M:AUTOCALL.UA1*]: [ENTER]

M:AUTOCALL.UA1 - Exists, ok to overwrite [Y/N*]: Y

Create another Autoscheduler file [Y/N*]: [ENTER]
```

```
*  VERIFYING FILES, BASIC WORKSPACE
SETTING SEQ/MET, BASIC WORKSPACE
AUTOSCHEDULER WORKFILE COMPATIBLE WITH SYSTEM

Start a sequence or a run [R*/S/Q]: [ENTER]

* RUN #    1      AUG  28, 1994  13:46:51
START
```

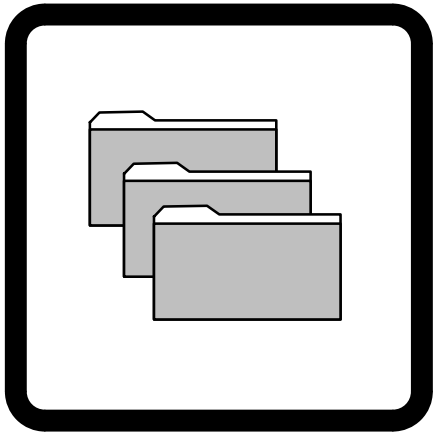



```
Closing signal file M:SIGNAL   .BNC
Storing processed peaks to M:Q1EB7A13.PRO

RUN#     1          AUG  28, 1994  13:46:51

SIGNAL FILE:  M:SIGNAL.BNC
PEAK FILE  :  M:Q1EB7A13.PRO
AREA%
      RT      AREA  TYPE  WIDTH      AREA%
    .082     36970    B     .03     81.8568
    .173         2    V     9            9
              79010    V     .04     18.1431
TOTAL AREA=448712      B     2            1
MUL FACTOR=1.0000E+00
```

```
Signal file M:SIGNAL.BNC renamed M:TESTA001.BNC
Processed peak file M:Q1EC06E9.PRO renamed M:TESTA001.PRO

==============       END OF RUN       ==============
```

4

Managing Files



In this chapter....

The File Manager Program

The File Manager program is used to manage disk files. It expands the file management capabilities of the integrator by allowing group-selected and directory-wide file operations.

You can perform the following file operations:

- Copy file(s)
- Delete file(s)
- List the contents of a directory
- Format a disk
- List a file's contents
- Rename files(s)

The File Manager operations will only work with the 19 valid file types listed in chapter 7 of the HP 3395 and HP 3396 *Operating Manuals*.

The Wildcard Character

The wildcard character (*) cannot be used in a file name extension.

Starting the File Manager Program

1. **Press function key [1] to start the File Manager program.**

   The program prints the following message:

```
PLEASE WAIT LOADING PROGRAM ...
```

   After a short pause, the program prints the following header information:

```
WELCOME TO THE HP 3396 FILE MANAGER PROGRAM (Rev. B.01.00)


AT ANY '?' or FM> PROMPT:          'Q' [ENTER] Quits
                                   'S' [ENTER] Starts Over
                                   '+' Return to FM> Prompt


FILE MANAGER MODE:

FM>
```

2. **At this point, the File Manager program is ready for you to enter a command.**

Entering Commands

The File Manager dialog has three modes of parameter entry:

- Syntax Mode
- Prompt Mode

Syntax Mode

In the syntax mode, you enter the entire command string, including all parameters and options, before pressing **[ENTER]**.

```
FM> COPY M:TEST7.BNC  M:T7_3_11.BNC/O
```

The examples in this section will be in syntax mode.

Prompt Mode

In the prompt mode, you enter only the command. When you press **[ENTER]**, the File Manager program prompts you for each parameter and option for that command.

```
FM> CO
SOURCE                      [M:*] : ? M:
FILENAME(S)                   [*] : ? TEST7
EXTENSION                           [*] : ? BNC
DESTINATION                 [M:*] : ? M:
NEW FILENAME(S)               [*] : ? T7_3_11
...  reading source file(s)
(Press space bar to stop)

COPY M:TEST7   .BNC   TO   M:T7_3_11.BNC     OK!

FM>
```

Changing the Default Directory

To change the default directory, type the disk designator of the desired directory.

```
FM> M:

FM> E:
```

CO{**PY**} {*d:*} {*filename1.ext*} {,} {*e:*} {*filename2*} {*/O*}

COPY copies a file from the source drive (*d:*) to a destination drive (*e:*).

d:	The drive that contains the source file(s).
e:	The drive to receive the destination file(s).
filename1	The source file to be copied.
, or space	A separator: "," and " " are both valid.
filename2	The new filename for the destination file (optional).
.ext	The file type extension.
/O	An option to overwrite files having the same name. The file type extension must be specified or the /O option will not work.

When the drive is not specified, the default drive is used.

The asterisk (*) is the wildcard character. For group-selected and directory-wide COPY operations, you can substitute the asterisk for *filename1*, *filename2*, and *ext*.

Example 1

To copy all files from drive M: to drive B:

```
FM> COPY M:*.*  B:*
```

Example 2

To copy all processed peak files (`.PRO`) from the M disk to drive M:

```
FM> COPY M:*.PRO  M:*
```

Example 3

To copy the signal data file (SIGNAL.BNC) from the M disk to M: and rename it (`RW_03_13_BNC`):

```
FM> COPY M:SIGNAL.BNC  M:RW_03_13.BNC
```

Example 4

To copy all of the files on the M disk that begin with TEST to drive M:

```
FM> COPY M:TEST*.PRO  M:*
```

Example 5

To copy the signal file (SIGNAL.BNC) from the default disk, in this case M, to drive M: and rename it (TESTRUN.BNC).

```
FM> COPY SIGNAL.BNC  M:TESTRUN.BNC
WARNING: DUPLICATE FILE NAME
OVERWRITE                    [Y/N*/Y!]: ? Y [ENTER]
COPY    M:SIGNAL    .BNC    TO    M:TESTRUN.BNC      OK!
FM>
```

Note: Selecting `Y!` will overwrite all the files specified with a wildcard without prompting for each individual file.

DE{LETE} {*d:*} {*filename.ext*} {*/O*}
ERASE {*d:*} {*filename.ext*} {*/O*}
PURGE {*d:*} {*filename.ext*} {*/O*}

This command deletes the specified file(s) from the specified disk.

d:	The drive that contains the file(s). When the drive is not specified, the default drive is used.
filename	The filename(s) to be deleted.
.ext	The file type extension.
/O	An option to delete files without verification. The file type extension must be specified or the /O option will not work.

The asterisk (*) is the wildcard character. For group-selected and directory-wide DEL operations, you can substitute the asterisk for *filename* and *ext*. If the filename contains an asterisk, multiple files are deleted.

Example 1

To delete a single file:

```
FM> DEL M:DEFAULT.MET

...  reading source file(s)
DELETE FILE  M:DEFAULT  .MET    OK!

FM>
```

Example 2

To delete, *with* verification, all the processed files on the M disk beginning with the letter Q, type the following:

```
FM> DEL M:Q*.PRO

...  reading source file(s)
DELETE FILE   M:Q1E23B7A.PRO   ARE YOU SURE   [Y/*N]: Y
DELETE FILE   M:Q1E238B1.PRO   ARE YOU SURE   [Y/*N]: Y

FM>
```

Example 3

To delete, *without* verification, all the processed files on the M disk beginning with the letter Q, type the following:

```
FM> DEL M:Q*.PRO/O

...  reading source file(s)
(Press space bar to stop)
DELETE FILE   M:Q1E23B7A.PRO     OK!
DELETE FILE   M:Q1E238B1.PRO     OK!

FM>
```

Example 4

To delete all files on the M: drive, type the following:

```
FM> DEL M:*.*

...  reading source file(s)
DELETE FILE   M:TEST    .BAS   ARE YOU SURE   [Y/*N]: Y
DELETE FILE   M:OLDFILES.BAS   ARE YOU SURE   [Y/*N]: Y
DELETE FILE   M:NEWFILES.BAS   ARE YOU SURE   [Y/*N]: Y

FM>
```

DI{RECTORY} {*d:*} {*filename.ext*} {*/W*}

This command prints the directory of files for the specified drive.

The information provided by DIRECTORY consists of the complete name of the file, the length of the file in bytes, and the date and time of day it was created. Following this are the quantities of free disk space.

d:	The drive that contains the file(s). When the drive is not specified, the default drive is used.
filename	The filename(s) to be displayed.
.ext	The file type extension.
/W	An option to list only the file names

The asterisk (*) is the wildcard character. For group-selected and directory-wide DIR operations, you can substitute the asterisk for *filename* and *ext*.

Example 1

To obtain a directory list of all files on a specific drive:

```
FM> DIR M:

(Press space bar to stop)
...  getting directory

VOLUME NAME: MDISK                    DRIVE:  M
DATE: AUG  2, 1994     11:20:53

   FILE NAME           LENGTH    CREATED/VERSION
SIGNAL  .BNC              512  12/11/89 10:10:42
Q1EA8C22.PRO              512  12/11/89 10:21:02
Q1EA6F64.PRO              512  12/11/89 10:33:21

                  USED           FREE        MAX
FILES                3             27         30
BYTES             1536          81664      83968
```

Example 2

To print information about a single file:

```
FM> DIR M:SIGNAL.BNC

(Press space bar to stop)
...  getting directory

VOLUME NAME: MDISK                    DRIVE:  M
DATE: AUG  2, 1994      11:24:53

   FILE NAME            LENGTH    CREATED/VERSION
SIGNAL  .BNC               512  12/11/89 10:10:42

1 FILE(S)                   81664 BYTES FREE

FM>
```

Example 3

To print information about a group of files:

```
FM> DIR M:*.*/W

(Press space bar to stop)
...  getting directory

VOLUME NAME: MDISK                    DRIVE:  M
DATE: AUG  2, 1994       11:28:23

FILE NAMES:
SIGNAL  .BNC   Q1EA8C22.PRO   Q1EA6F64.PRO
Q1EA8FC11.PRO
Q1EA9A41.PRO   Q1EA9C66.PRO

                  USED           FREE        MAX
FILES                6             27         30
BYTES             1536          81664      83968

FM>
```

Note: In example 3, the /W option was used to list only the file names.

FORMAT {*d:*}, {*vol name*}, {*max files*}
INIT {*d:*}, {*vol name*}, {*max files*}

This command initializes a disk for use.

d: The drive containing the disk to be formatted. When the drive is not specified, the default drive is used.

vol name An optional volume name (six characters maximum).

The volume name must begin with a letter and cannot exceed six characters. The volume name can contain letters, numbers, and the underscore character.

max files The maximum number of files to be stored on the disk.

The normal range for the maximum number of files is from 64 to 256.

When the volume name and maximum files are not specified, they are defaulted to:

```
vol name = 'DISK'

max files = '256'
```

Example

To format the M: disk and assign the name RESULT to the disk volume:

```
FM> FORMAT M:,RESULT

YOU ARE GOING TO FORMAT DISK                  : 'D:'
VOLUME NAME                          : 'RESULT'
MAXIMUM OF FILES                     : '256'
CAUTION: ALL DATA WILL BE DESTROYED
ARE YOU SURE     [Y/*N]: Y
PLEASE WAIT FORMATTING IN PROGRESS

FM>
```

LI{ST} {*d:*} {*filename.ext*} {/*A*} {/*H*}

This command lists the contents of the specified file name.

d:	The drive that contains the file(s). When the drive is not specified, the default drive is used.
filename	The filename(s) to be listed.
.ext	The file type extension.
/*A*	An option to force listing to LIF ASCII format.
/*H*	An option to force listing to HEX format.

The LIST command prints the contents of these file types:

.BAA	BASIC ASCII file (LIF ASCII).
.RPT	Report file from an analytical run (LIF ASCII).
.RPA	Report file from an **ANALYZE** command (LIF ASCII).
.PRA	Processed peak file from an **ANALYZE** command (LIF ASCII).
.RAW	Unbunched signal data file.
.BNC	Bunched data file from an analytical run.
.BNA	Bunched data file from an **ANALYZE** command.
.UA1	The Autoscheduler file extension.
.UA2	(LIF ASCII)
.UA3	(LIF ASCII)

Note: LIF (Logical Interchange Format) ASCII files have a .BAA, .RPT, .RPA, .PRA, .UA1, .UA2, or .UA3 file extension.

Example 1

To list area slices of a .RAW, .BNC, or .BNA data file:

```
FM> LIST M:Q12A3488.BNA
INITIAL TIME (min)                  : ?  0.1
END TIME     (min)                  : ?  0.4
Searching for the first point
(Press space bar to stop)

Number            Time      Width             Height
                   min      sec/20                uv

    30          .1000000      4.00              166.00
    31          .1033333      4.00              163.00
    32          .1066667      4.00              160.00

Listing terminated by user request

FM>
```

Note: The limits for INITIAL TIME and END TIME are: 0.0005 </= T </= 6900. To list the data points in decending order, enter the larger of the two times first.

Example 2

To list the contents of an LIF ASCII file:

```
FM> LIST M:TEST.BAA
LIF TYPE = 1
CAUTION: PREMATURE HALT MAY DAMAGE DATA FILE
(Press space bar to stop)
10  ! TEST BASIC PROGRAM
20  PRINT 'TEST BASIC PROGRAM'
30  GOTO 20

# OF ASCII CHARACTER(S) : 69                LINE(S) : 3

FM>
```

Example 3

To list a file in Hex format:

```
FM> LIST M:TEST.BNC/H
(Press space bar to stop)

0300 03 81 03 81 01 CF 01 27 01 2A 03 1F 01 14 02 40   .......'*.....@
0310 01 67 09 41 02 48 03 E3 03 FA 01 39 08 53 01 FB
.g.A.H.....9.S..
0320 02 A9 03 38 02 B8 08 FE 02 6A 08 A1 00 B0 03 9A
...8.....j......
Listing terminated by user request

FM>
```

Example 4

To list an LIF ASCII file in Hex format:

```
FM> LIST M:AUTOCALL.UA1/H
CAUTION: PREMATURE HALT MAY DAMAGE DATAFILE
(Press space bar to stop)

0000 00 0C 20 41 55 54 4F 5F 53 49 4E 47 4C 45 4C 0D   ..
AUTO_SINGLE..
0010 20 4A 41 4E 20 20 31 2C 20 31 39 30 31 00 00 09    JAN  1,
1901...
0020 20 30 30 3A 34 36 3A 32 36 00 00 0F 20 4D 3A 41    00:46:26...
M:A
Listing terminated by user request
FM>
```

Note: Never use the [BREAK] key to stop the listing of an LIF ASCII file. This could cause the file to be damaged. Use the space bar to stop the listing.

RE{NAME} {*d:*} {*filename1.ext1*} {,} {*filename2.ext2*} {*/O*}

The RENAME command changes the name of *filename1* to *filename2*. The filenames must have the same type extension.

d:	The drive that contains the file(s). When the drive is not specified, the default drive is used.
filename1	The filename(s) to be renamed.
, or space	The separator: "," and " " are both valid.
filename2	The new filename(s).
.ext1	File type extension for filename1.
.ext2	File type extension for filename2. *.ext1* and *.ext2* must be the same.
/O	Option to overwrite files having the same name.

The asterisk (*) is the wildcard character. For group-selected and directory-wide REN operations, you can substitute the asterisk for *filename1* and *filename2*. If *filename1* and *filename2* are both wildcarded, *.ext* cannot also be wildcarded.

The renamed file remains on the designated drive (*d:*).

Example 1

To rename a file on the M: drive:

```
FM> RENAME M:RESULTS.RAW  DATAJL21.RAW
```

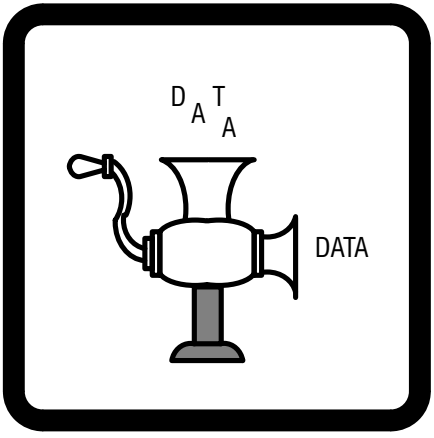
After the renaming operation, the file name `M:RESULTS.RAW` no longer exists.

Example 2

```
FM> REN
DEVICE                        [D:] : ? M:
FILENAME(S)                    [*] : ?
EXTENSION                      [*] : ?
NEW FIRST CHARACTER(S)         [*] : ? RSC_
...  reading source file(s)
(Press space bar to stop)

RENAME  M:EXECPRG1.BAA   TO   M:RSC_PRG1.BAA OK!
RENAME  M:EXECPRG2.BAA   TO   M:RSC_PRG2.BAA OK!
RENAME  M:EXECPRG3.BAA   TO   M:RSC_PRG3.BAA OK!

Operation terminated by user request.

FM>
```

5

Reprocessing Data Files



In this chapter....

The Batch Reprocessing Program

The Batch Reprocessing program reprocesses a group or sequence of data files that you specify. It reanalyzes them using the currently active method and information from either:

- The sample information table of a sequence
- The calibration options.

To reprocess calibrated data files, you must provide sample information, Sample Amount, ISTD Amount, etc., for each sample so the calculations can determine the sample's unknown amounts.

Reprocessing Data Files Using a Sample Information Table

A sample information table can also supply the sample information when reanalyzing data files. The calibration will be updated with information from the sample table.

Reprocess data files with a sample information table when you are:

- Reprocessing data files resulting from a sequence.
- Reanalyzing a set of data files using different sample and calibration parameters for each file.
- Reanalyzing the results of one sequence with the sample information (Sample Amount, ISTD Amount, etc.) from another sample table.
- Correcting mistakes present in the sample table of the original sequence and reanalyzing the sequence to obtain the correct results.

Reprocessing Data Files Using Calibration Option Parameters

You can reprocess data files using the sample information stored with the calibration, the calibration options. In this case, you are prompted to identify the calibration samples and the level of recalibration.

Reprocess data files with the calibration options when you are:

- Reprocessing a few data files.
- Using the same sample and calibration information for each file.
- Developing and optimizing a method.

Reprocessing Data With Calibration Option Parameters

To reprocess data files with the calibration options:

- Edit the calibration option parameters, if necessary.
- Start the Batch Reprocessing program.
- Select calibration options for sample information.

Editing the Calibration Options

Edit the calibration options, if necessary, to supply the desired sample information to be used during reprocessing. The calibration options are stored with the calibration in a method file. See chapter 6 of the HP 3395 or HP 3396 *Operating Manual* for more information about the calibration options.

```
*  LOAD  METH  M:METHNAME.MET  @
```

1. **Load the method file you intend to use for reprocessing data files.**

```
*  EDIT CALIB  @

1 = CALIB PROCEDURE
2 = RETENTION TIME WINDOWS
3 = TABLE ENTRIES
4 = PEAK GROUPS
5 = CALIB OPTIONS
```

2. **Press [EDIT] [CALIB] [ENTER] to access the calibration options selection.**

```
SECTION TO BE EDITED: 5 [ENTER]
```

3. **Enter 5 to edit the calibration options.**

Editing the Calibration Options (cont.)

```
CALIBRATION OPTIONS
RF of uncalibrated peaks [0.0000E+00 ]: [ENTER]
Replace calibration fit [Y/N*]: [ENTER]
Disable post-run RT update [Y/N*]: [ENTER]
ISTD peak #: 4
ISTD AMT [1.0000E+00 ]: 0.05
SAMPLE AMT [0.0000E+00 ]:2.0
MUL FACTOR [1.0000E+00 ]: [ENTER]
```

4. **Respond to the dialog by entering the Sample Amount, ISTD Amount, and Multiplier factor values to be used when reprocessing data files.**

```
*  STORE  METH  M:METHNAME.MET  @
```

5. **Store the method file with the edited calibration information.**

Starting the Batch Reprocessing Program

In this procedure, you will update the calibration information of the active method.

1. **Press function key [2], labelled Reproc. Data, to start the Batch Reprocessing program.**

```
WELCOME TO THE HP 3396 BATCH REPROCESSING PROGRAM (Rev.B.00.03)


At Any Prompt:         'Q' [ENTER] Quits
                       'S' [ENTER] Starts Over


Method file for reprocessing [Current active*]: [ENTER]
```

2. **Enter the name of the method being used to reprocess the data files or press [ENTER] to accept the current active method.**

   This method contains the integration and calibration parameters that control how the data files are reprocessed. It will be loaded into the active workspace.

   The method also contains the calibration information. When reprocessing is complete, you can store the updated calibration information from recalibrations by saving the current method after the files are reprocessed.

```
Calibration file for reprocessing [Current active*]: [ENTER]
```

3. **Press [ENTER] to select the calibration information of the active method.**

   When you choose the calibration information of the active method:

   - The updated calibration information is written to the currently active workspace.
   - To save this information for future recall, you must save the current method or calibration.

   At this prompt, you could supply the name of a calibration file to use instead of the method's calibration information. See "Specifying a Calibration File for Reprocessing."

Select Calibration Options for Sample Information

```
Use Sequence Sample Table for reprocessing [Y*/N]: N
```

4. **Enter N to select the calibration options to supply the sample information for reprocessing.**

   When you choose *not* to use the sample table, the sample information in the calibration options of the specified calibration file will be used to reprocess the data files. It overwrites the calibration options information of the current calibration information. The sample information includes the Sample Amount, ISTD Amount, and Multiplier factor.

   By editing the calibration information, and changing the Sample Amount, ISTD Amount, and Multiplier values, you reprocess the data files with the new sample information. See "Editing the Calibration Options" on page 5-4.

```
Enter the first signal data file name: M:FPD9D011.BNC
```

5. **Enter the first data file in the sequence you want to reprocess.**

```
Enter the last signal data file name: M:FPD9D014.BNC
```

6. **Enter the last data file in the sequence you want to reprocess.**

   The alphanumeric value of the last signal data filespec must exceed the first. In the example above, the last signal data filespec, `M:FPD9D014.BNC` has a greater alphanumeric value than the first signal data filespec `M:FPD9D011.BNC`.

```
At the RECALIB LEVEL # prompt for each file; push [ENTER] for
no recalibration, enter the level number to recalibrate, or
enter 'D' if you don't want to reanalyze the file.

VOLUME NAME: FPDDAT   DRIVE:A

  FILE #        FILE NAME       RECALIB LEVEL #
    1       FPD9D011.BNC          1
    2       FPD9D012.BNC
    3       FPD9D013.BNC          3
    4       FPD9D014.BNC          D

End of selected files
```

7. **For each calibration standard, enter the recalibration level to be updated with new calibration information during reprocessing. Press [ENTER] to identify an entry as an ordinary sample.**

   A recalibration updates the calibration information in the currently active method.

```
Are you ready to reprocess the selected data files [Y*N]: Y
```

8. **Enter Y if all of your entries are correct and you are ready to reprocess.**

   Enter **N** if you want to edit the list or make a correction to one or more of your entries before reprocessing.

   At this point, the program reanalyzes the signal data files using the sample and recalibration information from the sample table of the sequence you specified.

Example

```
WELCOME TO THE HP 3396 BATCH REPROCESSING PROGRAM (Rev.B.00.03)

At Any Prompt:         'Q' [ENTER] Quits
                       'S' [ENTER] Starts Over


Method file for reprocessing [Current active*]: [ENTER]

Calibration file for reprocessing [Current active*]: [ENTER]

Use the Sequence Sample Table for reprocessing [Y*/N]: N

Enter the first signal data file name: M:FPD9D011.BNC

Enter the last signal data file name: M:FPD9D014.BNC

At the RECALIB LEVEL # prompt for each file; push [ENTER] for
no recalibration, enter the level number to recalibrate, or
enter 'D' if you don't want to reanalyze the file.

0VOLUME NAME: FPDDAT   DRIVE:A

  FILE #         FILE NAME        RECALIB LEVEL #
    1       FPD9D011.BNC            1
    2       FPD9D012.BNC
    3       FPD9D013.BNC            3
    4       FPD9D014.BNC            D

End of selected files

Are you ready to reanalyze the selected files [Y*/N]: Y
```

At this point, the program reanalyzes the signal data files `M:FPD9D011.BNC` through `M:FPD9D014.BNC` in numerical order. The reports will be generated using the sample and recalibration information from the currently active method including the Sample Amount, ISTD Amount, and Multiplier. This is *not* the same as the analyze function, which uses the sample information stored with the data file.

Editing a File Entry

```
Are you ready to reprocess the selected data files [Y*/N]: N

To edit the file sequence, enter the file number to be edited. At the
RECALIB LEVEL # prompt, push [ENTER] for no calibration, enter the
correct level, or enter a 'D' if you do not want to analyze the file.
At 'File #' , an [ENTER] without a number ends editing.

  FILE #        FILE NAME       RECALIB LEVEL #
    3       FPD9D013.BNC           2
```

Reprocessing With Data From a Sequence Sample Table

Specify a sequence sample table that will supply the desired sample information during reprocessing. You can create a new sequence with the required information or edit and save an existing sequence.

To reprocess data files with sample table information, you must:

- Create and/or edit and save a sequence. Then supply the sequence sample table with the sample information to use for each data file being reprocessed.
- Start the Batch Reprocessing program.
- Enter the name of the sequence file.

Editing the Sequence Sample Table

If you are reprocessing calibrated data files produced by a sequence, use the *sequence sample table* to supply the sample information.

If necessary, edit the sample table of the sequence to which you intend to supply the sample information during reprocessing. See chapter 9 of the *HP 3395 or 3396 Integrator User's Manual* for more information about the sample information table.

```
*  LOAD  SEQ  M:SEQNAME.SEQ  @
```

1. **Load the sequence file to use for reprocessing data files.**

```
*  EDIT SEQ  @

1 = EQUILIBRATION TIME DELAY
2 = METHOD FILE SPECIFICATION
3 = SAMPLE INFORMATION TABLE
```

2. **Press [EDIT] [SEQ] [ENTER] to access the list of sequence sections to edit.**

```
SECTION TO BE EDITED: 3 [ENTER]
```

3. **Enter 3 to edit the sample information table.**

```
SAMPLE INFORMATION TABLE
BOTTLE OR RUN SAMPLE INDEXED [R/B*]: [ENTER]
```

4. **Enter the fist and last bottle (or run number) for the data files being reanalyzed.**

   The reprocessing program will attempt to match the bottle number (or run number when run indexing) of each data file being reprocessed to an index entry in the sample information table. If no match occurs, the sample information from the last successful index match is used.

```
BOTTLE # : 1 [ENTER]
ISTD AMT: 0.05 [ENTER]
SAMPLE AMT: 2.0 [ENTER]
MUL FACTOR: 1 [ENTER]
RECALIBRATION [Y/N*]: [ENTER]
NAME: STREAM1 [ENTER]
REPORT MEMO: [ENTER]
```

5. **Respond to the dialog by entering the ISTD Amount, Sample Amount, and Multiplier factor values to be used when reprocessing data files. You can also identify the recalibration level.**

```
BOTTLE # : 2 [ENTER]
ISTD AMT: 0.05 [ENTER]
SAMPLE AMT: 2.0 [ENTER]
MUL FACTOR: 1 [ENTER]
RECALIBRATION [Y/N*]: Y [ENTER]
LEVEL: 5 [ENTER]
NAME: CAL [ENTER]
REPORT MEMO: [ENTER]
```

6. **Enter the ISTD Amount, Sample Amount, etc., for each data file.**

```
BOTTLE # : [ENTER]
```

7. **Press [ENTER] for the BOTTLE # prompt to stop editing.**

```
*  STORE  METH  M:METHNAME.MET  @
```

8. **Store the sequence file with the edited sample table information.**

Starting the Batch Reprocessing Program

In this procedure, you will elect to update the calibration information of the active method.

1. **Press function key [2], labelled Reproc. Data, to start the Batch Reprocessing program.**

```
WELCOME TO THE HP 3396 BATCH REPROCESSING PROGRAM (Rev.B.00.01)


At Any Prompt:          'Q' [ENTER] Quits
                        'S' [ENTER] Starts Over


Method file for reprocessing [Current active*]: [ENTER]
```

2. **Enter the name of the method to use when reprocessing the data files.**

   This method contains the integration parameters that will control how the data files are reprocessed. It will be loaded into the active workspace.

   The method also contains the calibration information. When reprocessing is complete, you can store the updated calibration information from recalibrations by saving the current method after the files are reprocessed.

```
Calibration file for reprocessing [Current active*]: [ENTER]
```

3. **Press [ENTER] to select the calibration information of the active method.**

   When you choose the calibration information of the active method:

   - The updated calibration information is written to the currently active workspace.
   - To save this information for future recall, you must save the current method.

   At this prompt, you could supply the name of a calibration file to use instead of the method's calibration information. See "Specifying a Calibration File for Reprocessing."

```
Use the Sequence Sample Table for reprocessing [Y*/N]: Y
```

4. **Enter Y to select the sample table of a sequence you will specify as the source of sample information.**

```
Sequence file with Sample Table [ENTER = current active]: M:FPD3
```

5. **Enter the name of the sequence that contains the sample table you've chosen (and perhaps edited) .**

   If you do not supply the .SEQ file extension, the program does it for you.

   This sample table must contain the Sample Amount, ISTD Amount, etc., for each data file in the sequence being reanalyzed.

```
Enter the first signal data file name: M:FPD9D011.BNC
```

6. **Enter the first data file in the sequence you want to reprocess.**

   You must include the disk drive identifier and the file extension for the first signal data filespec.

```
Enter the last signal data file name: FPD9D014
```

7. **Enter the last data file in the sequence you want to reprocess.**

   If you do not supply the file extension, the program does it for you.

   The the disk drive identifier and the file extension for the second signal data filespec are optional.

   The alphanumeric value of the last signal data filespec must exceed the first. In the example above, the first signal data filespec, `M:FPD9D014.BNC` has a greater alphanumeric value than the last signal data filespec `M:FPD9D011.BNC`.

   At this point, the program reanalyzes the signal data files using the sample and recalibration information from the sample table of the sequence you specified.

Example

```
WELCOME TO THE HP 3396 BATCH REPROCESSING PROGRAM (Rev.B.00.01)


At Any Prompt:          'Q' [ENTER] Quits
                        'S' [ENTER] Starts Over


Method file for reprocessing [Current active*]: [ENTER]

Calibration file for reprocessing [Current active*]: [ENTER]

Use the Sequence Sample Table for reprocessing [Y*/N]: Y

Sequence file with Sample Table [Current active*]: M:FPD3

Enter the first signal data file name: M:FPD9D011.BNC

Enter the last signal data file name: M:FPD9D014.BNC
```

At this point, the program reanalyzes the signal data files `M:FPD9D011.BNC` through `M:FPD9D014.BNC` in numerical order using the sample and recalibration information from the sample table of the specified sequence.

Specifying a Calibration File for Reprocessing

Referring to step 3 on pg. 6 and step 3 on pg.13 in this chapter, you can specify a calibration file for reprocessing data files or an update after reprocessing.

```
Calibration file for reprocessing [Current active*]: [ENTER]
```

Enter the name of an optional calibration file to use instead of the calibration information of the active method.

When you specify a calibration file for reprocessing:

- The calibration file overwrites *all* of the method's calibration information, including the sample information specified in the calibration options. The new values of Sample Amount, ISTD Amount, and Multiplier factor are used to reprocess the data files.
- The calibration information of the currently active method is updated but not stored in the method file. *To incorporate the updated calibration in the method file, you must store the method.*

```
*  [STORE] [METH] "method file spec" [ENTER]
```

Enter a calibration file when you want to reprocess data files with a single method (one set of integration parameters) using a different calibration file.

Storing the Updated Calibration Information

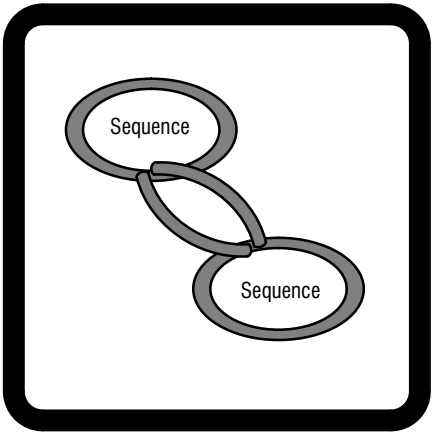
If you specified a calibration file for reprocessing:

- The specified calibration file will not be updated with new data when the signal data files are reprocessed. Only the currently active calibration is updated.
- To save the calibration information for future recall, save the calibration file or save the method file used for reprocessing.

  Storing this calibration file does not overwrite the calibration information in the method file, so the method file (and its calibration information) remains unchanged.

6

Chaining Sequences



In this chapter....

The Sequence Chaining Program

The Sequence Chaining program allows you to string several sequences together to form a "chain" of sequences. Each sequence in the chain may have different method and sequence table parameters to analyze different types of samples.

When you start the Sequence Chaining program:

1. The first sequence executes using its assigned method to analyze the samples. After each run, the postrun program (optional) executes.
2. The next sequence in the chain executes using its assigned method to analyze the samples. After each run, the postrun program (optional) executes.
3. And so on until the last sequence in the chain completes.

Before You Start

The Sequence Chaining program has the following requirements:

- The sequence and method files must already exist.
- The Autoscheduler files must already exist if you plan to schedule postrun programs. See chapter 2, "Scheduling Postrun Programs," for more information about the Autoscheduler.

How Calibration Information is Stored

When a sequence is finished or when the sequence loads a new method, the calibration table (in the active workspace) is copied into and stored with the method. Subsequent runs using this method will reflect the updated calibration information.

Running the Sequence Chaining Program

At power on, the default key assignment for the Sequence Chaining program is function key [6]. Unless you've changed the key assignment, pressing this key will start the Sequence Chaining dialog.

Pressing **[ENTER]** at any dialog prompt will input the default value (i.e., the currently active sequence, method, Autoscheduler file, or the default selection marked with an asterisk).

1. **Press function key [6], labelled Seq. Chain, to start the Sequence Chaining program.**

```
WELCOME TO THE HP 3396 SEQUENCE CHAINING PROGRAM (Rev.B.00.02)

  At Any Prompt:       'Q' [ENTER] Quits
                       'S' [ENTER] Starts Over

Copying M:AUTOCALL.UA1 to M:HPSEQTMP.UA1 ......


Sequence list already defined on Jun 1, 1990

Do you want to redefine [Y*/N]: [ENTER]
```

2. **If a sequence chain already exists, this prompt will be printed. Press [ENTER], or enter Y to create a new sequence chain.**

   If you enter **N,** the existing sequence chain will start.

Entering the Sequences to be Chained

The Sequence Chaining dialog prompts you to specify each sequence in the chain and its associated method. Enter the sequence and method files in the order you want them to be executed.

```
Enter Sequence file [X:XXXXXXXX.SEQ*]: M:SEQ1.SEQ
```

3. Enter the name of an existing sequence file to be the first in the sequence chaining list.

Enter the complete sequence file specifier including the disk drive and the file extension. If you do not supply the disk specifier, it defaults to M:.

Initially, the default sequence file name is `[X:XXXXXXXX.SEQ*]`. Once you enter a sequence file name, it becomes the default and appears as the default sequence file for subsequence sequence file prompts. For example, the entry made in this step, **M:SEQ1.SEQ**, becomes the default sequence file for the next sequence prompt. See step 6 below.

Note: If you enter a run indexed sequence, the following message is printed:

```
Enter Sequence file [X:XXXXXXXX.SEQ*]: M:SEQ1.SEQ
 SEQUENCE SPECIFIES RUN # INDEXING: FIRST RUN = nnn LAST RUN = nnn
 CAUTION : INTEGRATOR RUNNUM WIL BE SET TO nn AT START TIME


Enter Method file [X:XXXXXXXX.MET*]: M:METH1.MET
```

4. Enter the name of an existing method file to be assigned to the sequence in step 4.

Specify a method file with the complete method file specifier including the disk drive and the file extension. If you do not supply the disk drive specifier, the program defaults it to M:.

Initially, the default method file name is `[X:XXXXXXXX.MET*]`. Once you enter a method file name, it becomes the default and appears as the default method file for subsequence sequence file prompts. For example, the entry made in this step, **M:METH1.MET**, becomes the default method file for the next method prompt. See step 6 below.

To use the method specified in the sequence, press **[SPACE BAR] [ENTER]**.

```
Enter Autoscheduler file [X:XXXXXXXX.UA1*]: M:AUTOCALB.UA1
```

5. To schedule a postrun program to execute after each run of the sequence, enter the name of an existing Autoscheduler File.

You can assign an optional Autoscheduler file to any sequence in the chain for scheduling and executing postrun programs. Enter the complete file specifier including the disk drive and the file extension. If you do not supply the disk specifier, it defaults to M:.

Initially, the default file name is `[X:XXXXXXXX.UA1*]`. Once you enter an Autoscheduler file name, it becomes the default and appears as the default Autoscheduler file for subsequence prompts. For example, the entry made in this step, **M:AUTOCALB.UA1**, becomes the default Autoscheduler file for the next Autoscheduler prompt. See step 6 below.

To specified no Autoscheduler file for this sequence, press **[SPACE BAR] [ENTER]**.

Note: When the sequence chain runs, the Autoscheduler file for the sequence currently running is renamed to **M:AUTOCALL.UA1** so the Autoscheduler can use it. When the sequence is finished, the original Autoscheduler file name designated in the chain is returned. See "Creating and Storing Multiple Autoscheduler Files" in chapter 2 of this manual for more information.

```
Enter the Sequence file name [M:SEQ1.SEQ*]: M:SEQ2.SEQ
Enter the Method file name [M:METH1.MET*]: M:METH2.MET
Enter the Autoscheduler file name [M:AUTOCALB.UA1*]: M:AUTOANBL.UA1
```

6. Enter the next sequence, method, and Autoscheduler File (optional) for the next sequence in the chain.

Continue entering sequence, method, and Autoscheduler names until you've entered the last sequence in the chain.

Running the Samples

```
Enter the Sequence file name [X:XXXXXXXX.SEQ]: [SPACEBAR] [ENTER]
```

7. To start the sequence chain and run the samples, press the [SPACEBAR] followed by [ENTER] to quit the dialog and start the sequence chain.

When the sequence chain runs:

1. The first sequence in the chain is loaded.
2. Next, the method associated with the first sequence is loaded.
3. The first sequence executes, using the assigned method to run each vial specified in the sample information table.

   After each run, the optional postrun program executes (if specified).
4. The second sequence executes, using its assigned method to run each vial specified in the sample information table.
5. The optional postrun program (if specified) for the second sequence executes after each run.
6. The program stops when all of the samples in the last sequence in the chain have been run.

Stopping a Sequence Chain

To stop the sequence chain in progress, press the **[BREAK]** key.

Restarting a Sequence Chain

To restart the sequence chain, press key 6. After the program header is printed, the restart prompt appears.

```
Continue from [BREAK] or Start Over [C*/S]:
```

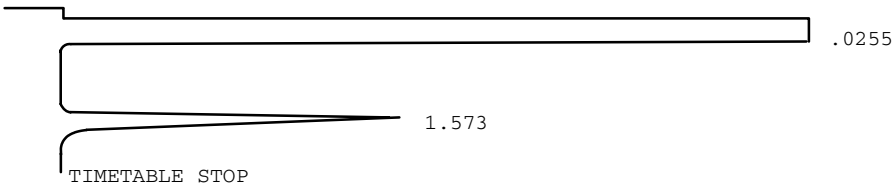
- Press **C** to continue the sequence chain at the point you pressed the [BREAK] key.
- Press **S** to restart the Sequence Chain program from scratch and build a new chaining workfile.

Looking Over the Results

The results of a two sequence chain are shown on the next few pages. When this sequence chain was prepared, sequence # 1 was not assigned an Autoscheduler file. Sequence # 2 was assigned an Autoscheduler file that runs the Autonaming postrun program.

Sequence Number 1, Run Number 1

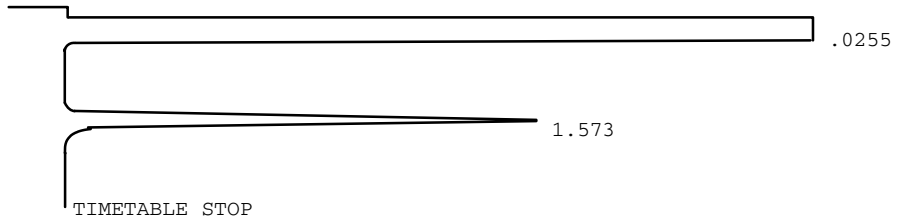
```
Loading Sequence.....M:SEQ1.SEQ
Loading Method......M:METH1.MET


Waiting for System Readiness
RUN #   1     JAN  1, 1995  02:44:10
START
```




```
Closing Signal File M:Q1EB8752.BNC
Storing processed peaks to M:Q1EB8752.PRO


RUN #   1           JAN  1, 1995  02:44:10


SAMPLE NAME: 121589T1         SAMPLE#   1


SIGNAL FILE: M:Q1EB8752.BNC
PEAK FILE  : M:Q1EB8752.PRO
ESTD%-AREA

      RT      AREA TYPE    CAL#       AMOUNT
      .255  262736   BB      1         900.122
      .755   26239   BH                 99.877
                         2R
 TOTAL AREA=288975
 MUL FACTOR=1.0000E+00
```

Sequence Number 1, Run Number 2

```
RUN #   2     JAN  1, 1995  02:48:21
START
```



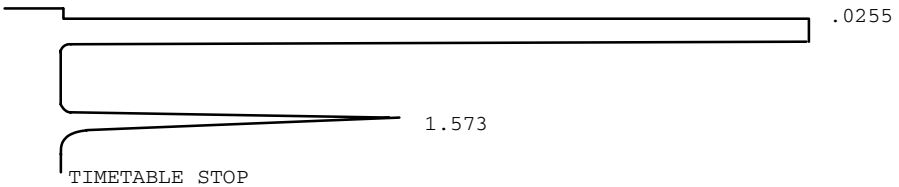

```
Closing Signal File M:Q1EB8803.BNC
Storing processed peaks to M:Q1EB8803.PRO


   RUN #   2            JAN  1, 1995  02:48:21


   SAMPLE NAME: 121589T2          SAMPLE#   2


   SIGNAL FILE: M:Q1EB8803.BNC
   PEAK FILE  : M:Q1EB8803.PRO
   ESTD%-AREA

        RT      AREA  TYPE    CAL#       AMOUNT
        .255   26273   BB       1        899.988
        .755       6   BH                193.752
              51142         2R

   TOTAL AREA=313878
   MUL FACTOR=1.0000E+00
```

The Sequence Chaining program will run all of the samples in sequence number 1 and then chain to sequence number 2.

Sequence # 2 was assigned an Autoscheduler file that runs the Autonaming program.

Sequence Number 2, Run Number 1

```
VERIFYING SINGLE/DUAL CHANNEL, BASIC WORKSPACE
INSTALLING/VERIFYING INET PARAMETERS
SEQUENCE FILE -> REALTIME = M:SEQ2.SEQ
METHOD FILE -> REALTIME = M:METH2.MET
SETTING BASIC SPACE
AUTOSCHEDULER WORKFILE COMPATIBLE WITH SYSTEM


 Waiting for System Readiness
RUN #   1      JAN  1, 1995  02:54:13
START
```




```
Closing Signal File M:Q1EB8832.BNC
Storing processed peaks to M:Q1EB8832.PRO


RUN #   1           JAN  1, 1995  02:54:13

SAMPLE NAME: 122689T1          SAMPLE#   1


SIGNAL FILE: M:Q1EB8832.BNC
PEAK FILE   : M:Q1EB8832.PRO
ESTD%-AREA

      RT     AREA TYPE    CAL#       AMOUNT
      .255  262736   BB      1        900.122
      .755   26243   BH                99.997
                           2R
 TOTAL AREA=288979
 MUL FACTOR=1.0000E+00



 Signal file M:Q1EB8832.BNC renamed M:TESTA001.BNC

 Processed peak file M:Q1EB8832.PRO renamed M:TESTA001.PRO
```

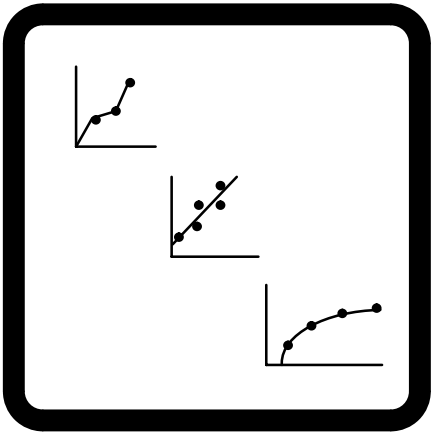
Sequence Number 2, Run Number 2

```
RUN #   2     JAN  1, 1995  03:08:23
START
                                                                .0255
                                          1.573
TIMETABLE STOP


Closing Signal File M:Q1EB8855.BNC
Storing processed peaks to M:Q1EB8855.PRO

RUN #   2            JAN  1, 1995  03:08:23

SAMPLE NAME: 122689T2          SAMPLE#   2

SIGNAL FILE: M:Q1EB8855.BNC
PEAK FILE  : M:Q1EB8855.PRO
ESTD%-AREA

      RT     AREA TYPE   CAL#       AMOUNT
      .255  26274   BB     1         899.997
      .755      5   BH               193.811
           51232        2R
TOTAL AREA=313977
MUL FACTOR=1.0000E+00

Copying M:AUTOCALL.UA3 to M:AUTOCALL.UA1


END OF PROGRAM



Signal file M:Q1EB8855.BNC renamed M:TESTA001.BNC

Processed peak file M:Q1EB8855.PRO renamed M:TESTA001.PRO

==============       END OF RUN       ==============
```

The program stops when all of the samples have been run in sequence number 2.

7

Plotting a Calibration Curve



In this chapter....

The Calibration Curve Plotting Program

The curve plotting program plots the response curves of calibrated peaks from either a method or calibration file. The calculations used to produce the plots are identical to the integrator's internal curve fitting calculations. Absolute amounts are used in each case.

You can plot calibration data from either:

- A method or calibration file that you specify.
- The current active method.

When plotting a calibration curve, you can:

- Specify the plotting parameters.
- Use the plotting program defaults.

Using the Calibration Curve Plotting Program

When the integrator is first switched on, the default key assignment for plot calibration curve is function key [3].

Unless you've changed the key assignment, pressing this key will start the plot calibration curve dialog.

Starting the Program

1. **Press function key [3], labelled Plot Calib., to start the plot calibration curve program.**

```
WELCOME TO THE HP 3396 CALIBRATION CURVE PLOTTING PROGRAM Rev. B.00.06

At any prompt:        'Q' [ENTER] Quits
                      'S' [ENTER] Starts Over
```

Specifying the Method or a Calibration File to Plot

```
Load which method or calib. file [Current active*]: [ENTER]
```

2. **Enter the name of the calibrated method or calibration file you wish to plot, or press [ENTER] to specify the current active method.**

   Press **[ENTER]** to plot the peaks in the current active method, or type the name of the method or calibration file to plot.

   You must include the device designator when you specify a file. Use the following format:

   ```
   M:TESTMETH.MET
   ```

   Only calibrated methods can be plotted. If you specify an uncalibrated method, an error message is printed:

   ```
   This Method Contains NO Calibrated Peaks !
   ```

Selecting the Peak(s) to Plot

```
Plot the calibration curve for which CAL# [All*]: 1
```

3. **Enter the calibration number, in the calibration table, of the peak to plot, or enter A to plot all of the peaks in the calibration.**

   Press **[ENTER]** if you want to plot all of the calibrated peaks.

   Each plot takes approximately three minutes to complete, so be sure you really want to plot all of the peaks.

   - If you choose to plot all calibrated peaks, then the plotting will use default plotting parameters from the current active method.
   - If the method is a single-level calibration, then the response factor for each calibrated peak will be printed but no plot will be drawn.

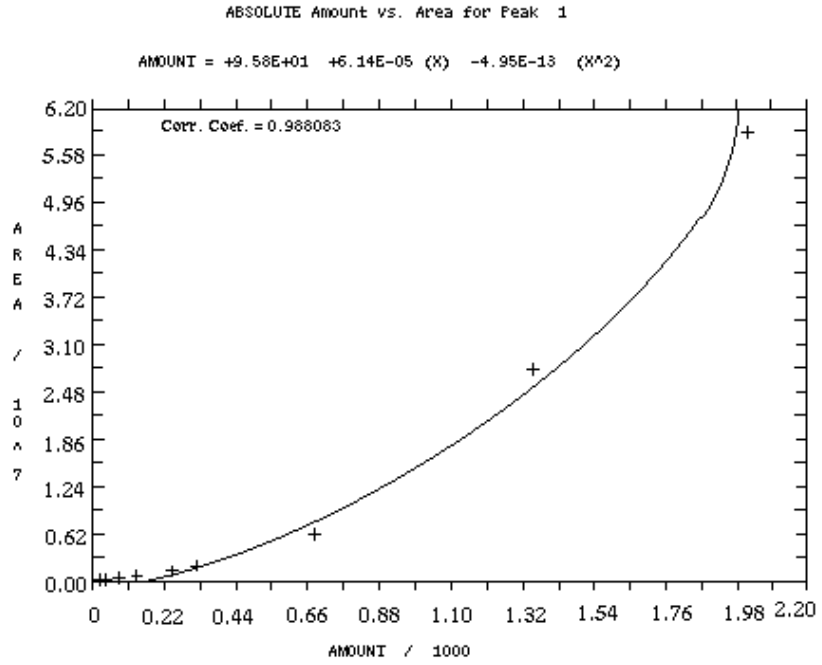
Using the Default Plotting Parameters

The next prompt determines which plotting parameters to use. You can use the program's default plotting parameters or choose your own plotting parameters.

```
Choose plotting parameters or plot with defaults [C/D*]: [ENTER]
```

4. Press [ENTER] to use the default plotting parameters.

Select the defaults to plot peaks using the curve type specified in the current active method (point-to-point, linear, or non-linear) and the program's default plotting parameters. The plotting begins immediately.



The horizontal axis is in units of absolute amount and the vertical axis is in units of area counts.

```
Plot additional peaks [Y*/N]:
```

5. When the curve plotting is finished, press [ENTER] to plot additional calibration numbers (calibrated peaks), or enter N to exit the program.

Choosing Plot Parameters

You can investigate alternate calibration curve plots by specifying different plot parameters for the same peak.

```
Choose plotting parameters or plot with defaults [C/D*]: C
```

4. Enter C to choose individual plotting parameters.

At this point, instead of plotting, the program prompts you for the plotting parameters.

- The curve fit parameter (point-point, linear, or non-linear)
- The maximum value for the area axis.

```
Enter maximum for area axis [AUTOSCALE*]: 60000000
```

5. Enter a maximum value for the area axis or press [ENTER] to select Autoscale.

- The maximum allowable entry for the area axis is 1E15.
- If you enter a value for the maximum area axis, the actual scaling used for the plot is twice the maximum calibrated area.

If your entry for the maximum area axis exceeds one of these limits, it will be ignored and you will be reprompted for a value within the allowable range.

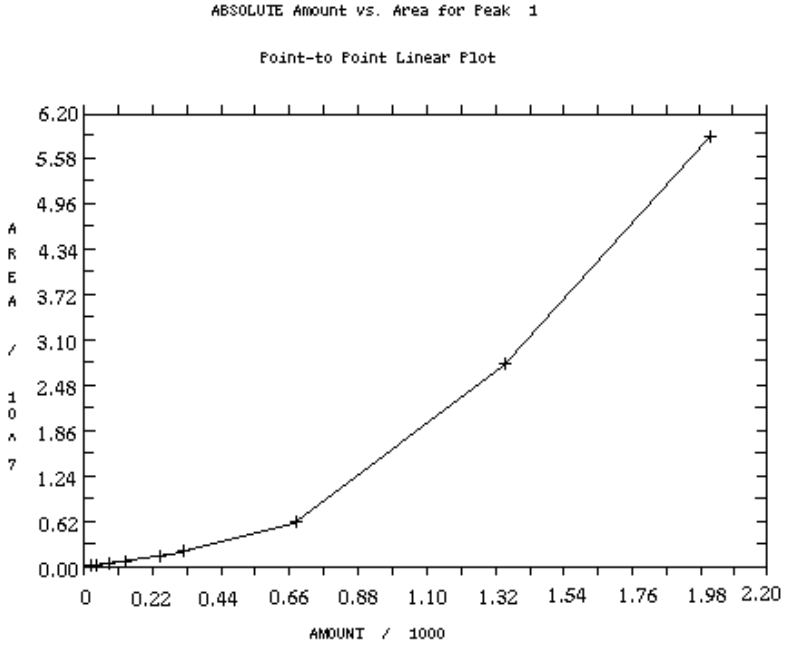
If you select Autoscale, the maximum scaling used is the largest calibrated area and amount plus five percent.

```
Select point-point, linear, or non-linear curve fit [P*/L/N]:
```

6. Press [ENTER] to select the default, a point-to-point curve fit, or enter an L or an N to select an alternative curve fit.

Type an **L** for a linear plot or **N** or a nonlinear plot.

At this point the plotting begins. The resulting plot appears on the next page.




```
Plot additional peaks [Y*/N]: N
```

7. Press [ENTER] to plot additional peaks, or enter N to stop plotting and exit the program.

If you choose to plot additional peaks, the programs returns to the following prompt:

```
Plot the calibration curve for which CAL# [All*]:
```

Plotting ISTD Peaks

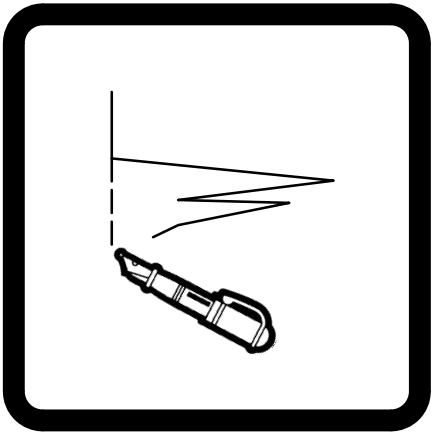
The integrator uses relative response factors when it calculates the amounts using internal standard.

Since the curve plotting program uses absolute amounts to produce the plots, plots of internal standard peaks will show the response factors for internal standard as independent absolute factors. This lets you verify the linearity of the internal standard.

The calibration table carries absolute response factors, but the calibration curves themselves are relative to the internal standard.

8

Plotting a Baseline



In this chapter....

The Baseline Program

When you make a run, the integrator produces a chromatogram and a report that are used to identify or characterize the sample. The accuracy of the report depends on how the chromatographic baseline is constructed. This means that while developing a method, you must constantly examine the baseline to ensure that the integration is being performed properly.

Practically speaking, however, examining and interpreting each baseline in detail is a tedious, time consuming process.

Often, a better way to check the baseline construction is to let the Baseline program replot the chromatogram and automatically draw in the baseline for you, showing you exactly how the baseline was constructed.

How the Baseline Program Works

The Baseline program replots the chromatogram obtained during an analysis and draws in the baseline, showing exactly how the integration was done.

The Baseline program can be automated as a postrun program or run manually by pressing the key labelled **Baseline**.

When the chromatogram is replotted:

- The signal and processed peak files of the original chromatogram supply the chromatographic and baseline information.
- Method parameters determine the replotted chromatogram's appearance.

Note: The Baseline program cannot access signal and processed peak files from a host device (drive specifier H:).

Running the Baseline Program with the Autoscheduler

When you schedule Baseline as a postrun program, it executes immediately after the run. At the end of the run, the program replots the chro-matogram and draws the baseline using the *current* method, signal, and processed peak files.

Running the Baseline Program Manually

Press function key [8] to run the the Baseline program manually.

When the program is run manually, the interactive dialog prompts you for the method, signal file, processed peak files, and other parameters. When you have supplied all of the required information, the program replots the chromatogram and draws the baseline using the parameters *that you specified* in the dialog.

Running the Baseline Program as an Autocall Program

You can run the Baseline program as an Autocall program by assigning `E:BASELINE.BAS` to function key [0].

Preparing to Run the Baseline Program

To replot a chromatogram with its baseline, the run must store both the signal file (bunched or raw data) and the processed peak file of the original chromatogram. If necessary, select option 2, enter the appropriate run data storage parameters for the method being used, and save the method.

Example

The example below shows how to set up the data storage parameters for the original chromatogram. In this instance, both the bunched signal file and the processed peak file will be stored on the M disk.

[OP()] [2] [ENTER]

```
RUN DATA STORAGE
Store signal data [Y/N*]: Y [ENTER]
Device [M*]: [ENTER]
Bunched or raw data [B/R*]: B [ENTER]
Store processed peaks [Y/N*]: Y [ENTER]
Device [M*]: [ENTER]
```

* Current default values are identified by an asterisk.

In most instances, you should store bunched data because:

- Bunched data uses less memory per analysis than raw data.
- The Baseline program must reanalyze a raw signal file to produce a filtered (bunched) file for a filtered plot. That takes more time than simply replotting a bunched signal file.

Running the Baseline Program with the Autoscheduler

To run the Baseline program using the Autoscheduler, follow the steps below.

1. **Load the method for this analysis.**
2. **Press function key [9] to start the Autoscheduler program.**
3. **Press [ENTER] to accept the current dialog selections until the heading** `'SCHEDULING AUTOCALL PROGRAMS'` **is printed.**
4. **To schedule the Baseline program as a postrun program, respond to the dialog prompts as follows:**

```
SCHEDULING AUTOCALL PROGRAMS
----------------------------
Schedule postrun AUTOCALL programs [Y/N*]: Y

AUTOCALL filespec : E:BASELINE.BAS

AUTOCALL filespec : [ENTER]

Store the current Autoscheduler file ('N' re-edits) [Y/N*] Y

Enter autoscheduler file name [M:AUTOCALL.UA1*]: [ENTER]

M:AUTOCALL.UA1 - Exists, ok to overwrite [Y/N*]: Y

Create another Autoscheduler file [Y/N*]: [ENTER]

DONE
```

Note: If you are also scheduling the Autoname program, Autoname must be scheduled as the first AUTOCALL filespec.

Starting the Run or Sequence

After you have prepared the Autoscheduler file, you can start the autoscheduled run or sequence.

You *must* use function key [0] to start the run or sequence. If you start a run by pressing the **[START]** button on the integrator, GC, etc., the run will start, but the postrun programs will *not* execute.

After each run, the postrun program(s) specified in the `M:AUTOCALL.UA1` Autoscheduler file will execute. In this example, the postrun program is the Baseline application program.

1. **Press function key [0] to initiate the START dialog.**

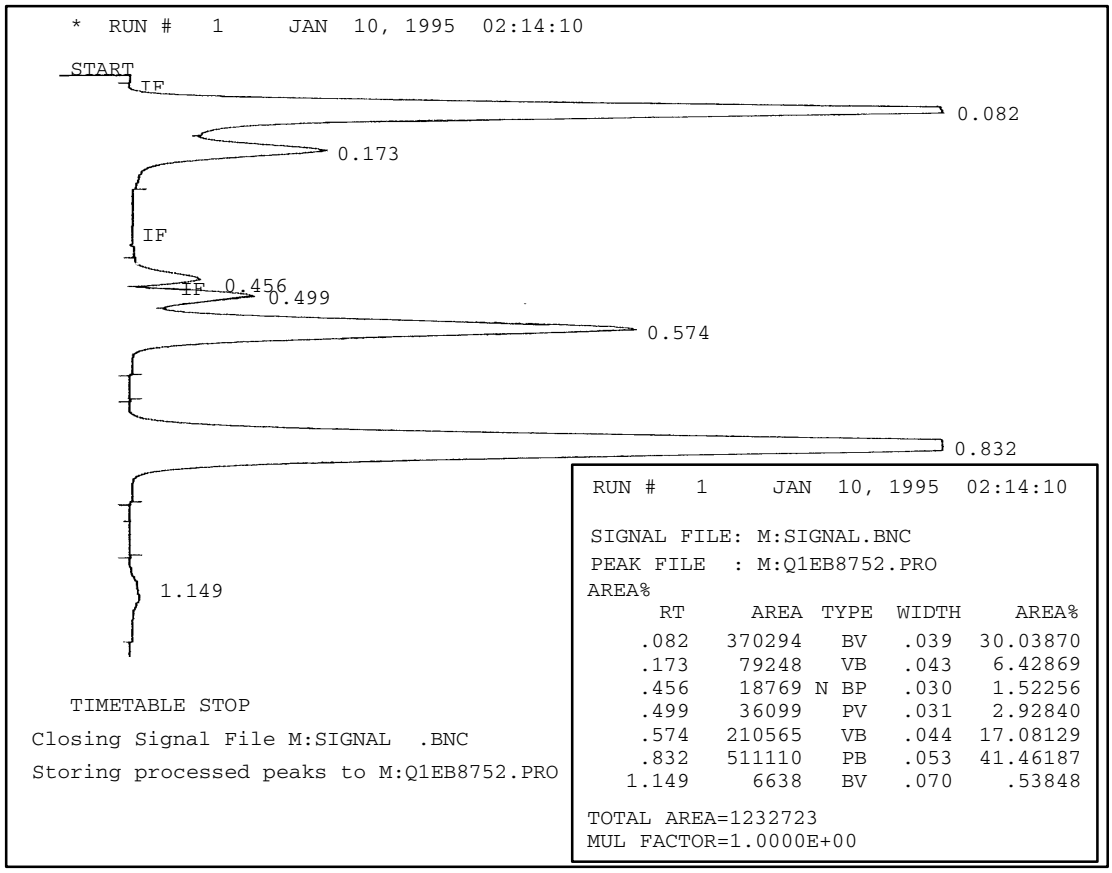
   When you press key [0], labelled **Auto Start**, a program called `AUTO_TOP` performs a number of prerun tests to check program space to ensure that the postrun programs will execute. Each test will print a confirming message or an error message at its conclusion. If an error message is printed, see the Appendix for error recovery information.

```
*  VERIFYING FILES, BASIC WORKSPACE
SETTING SEQ/MET, BASIC WORKSPACE
AUTOSCHEDULER WORKFILE COMPATIBLE WITH SYSTEM
```

   When everything checks okay, you are prompted to *start the run or sequence*.

```
Start a sequence or a run or quit [R*/S/Q]: [ENTER]
```

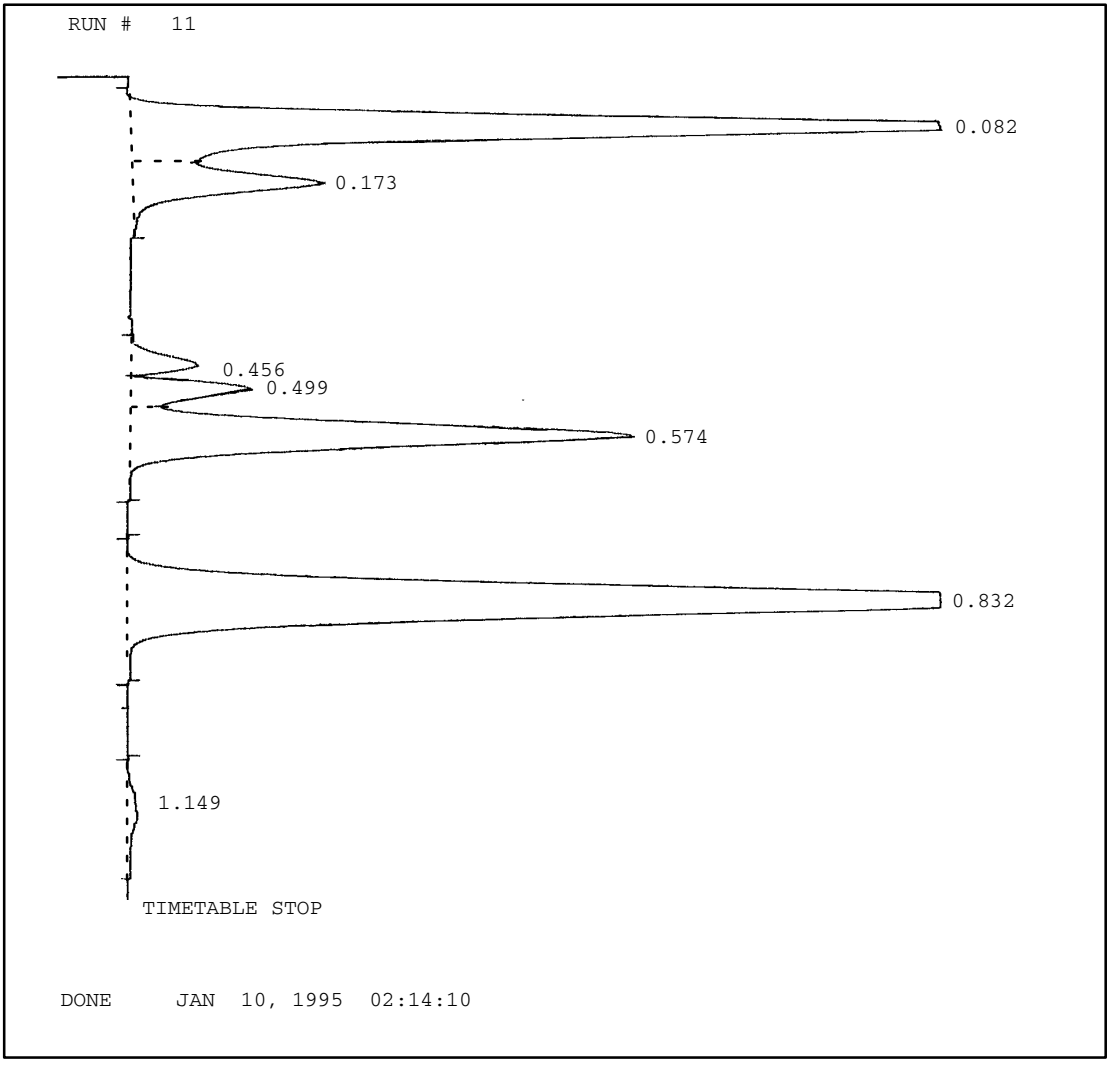
2. **Press [ENTER] to start the run.**



The Original Chromatogram.

This is the chromatogram resulting from the original run. The Baseline program replots this chromatogram and draws in the baseline based on the integration parameters. The replotted chromatogram and the baseline appear on the following page.

This is the chromatogram replotted by the Baseline program.



The Replotted Chromatogram with the Baseline Drawn.

The replotted chromatogram on the previous page has the following characteristics:

- The retention times will be annotated unless disabled with integration function 7 (IF 7).
- Chart annotations for ZE, AT, CS, AR, PW, TH, IF, and ^ZE will appear on the replot as determined by the method.
- Only integration functions 7, 8, 11, and 12 of the method will *affect* the replot. Integration event 13 is forced on during the replot to display the result of functions 11 or 12 when they are used.

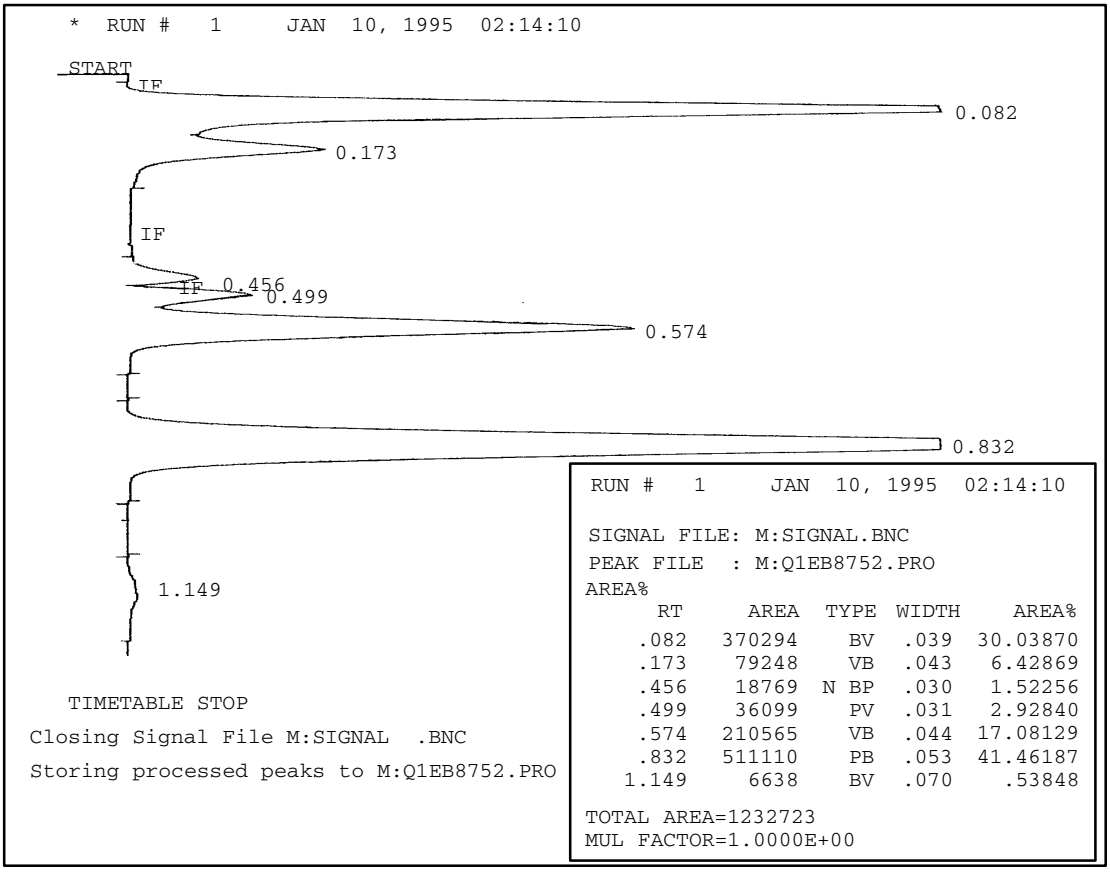
Running the Baseline Program Manually

You can run the Baseline program manually, using the interactive dialog. Before starting the Baseline program, you have to store some peak and baseline information by running an analysis.

1. **Check that the run data storage parameters are properly set with option 2 to store processed peak and signal files.**
   See page 8-4 about the run storage options.

2. **Run your analysis.**

   The result files will contain the processed peak and signal files for the Baseline program to replot.



The Original Chromatogram.

Starting the Interactive Dialog

3. Press function key [8] to start the postrun replot program.

```
WELCOME TO THE HP 3396 CHROMATOGRAM REPLOTTING PROGRAM (Rev. A.00.13)


At any prompt:      'Q'[ENTER] QUITS
                    'S'[ENTER] STARTS OVER
```

Plotting with the Current Method and Signal Files

```
Use the current method and data files for the replot [Y*/N]: Y
```

4a. Type Y or press [ENTER] to select the current method and data files.

```
PROCESSING
```

(See page 8-17 for the replotted chromatogram.)

Specifying the Signal Files to Plot

```
Use the current method and data files for the replot [Y*/N]: N
```

4b. Enter N to enter the dialog for independently specifying the method file and/or the signal files.

```
Signal file for the replot [M:TESTA001.BNC]: M:RUN12.BNC
```

5. Enter the name of the signal file from a previous analysis.

If run data has been stored, the current signal file name from the last run is shown in the brackets. Pressing **[ENTER]** without specifying a signal would enter the current signal file.

```
Processed peak file for the replot [M:TESTA001.PRO]: M:Q7BCB1C1.PRO
```

6. Enter the name of the processed peak file from a previous analysis that you want to replot.

If run data has been stored, the current processed peak file from the last run is automatically shown in brackets. Press **[ENTER]** to replot the current processed peak file.

The signal file and processed peak files must both come from the same run.

```
Start time for the replotted chromatogram [0.0]:
```

7. Press [ENTER] to accept the current value of 0.0 minutes as the start time for the replot.

```
Stop time for the replotted chromatogram [3.27]:
```

8. Enter the desired stop time for the replot.

The value in brackets is the end-of-signal time from the last run.

By selecting appropriate values for start and stop times, you can select only the portion of the replotted chromatogram and baseline you wish to appear.

```
Use the current method for plotting parameters [Y*/N]:
```

9a. Press [ENTER] to accept the current selection, Y.

```
PROCESSING
```

(See page 8-17 for the replotted chromatogram.)

Specifying a Method for the Plotting Parameters

```
Use the current method for plotting parameters [Y*/N]: N
```

9b. Enter N to indicate that you do not want to use the current method.

```
Specify a method file [Y*/N]: Y
```

10a. Press [ENTER] to accept the current selection, Y.

```
Method file [M:CURRENT.MET]: M:OLDMETH.MET
```

11a. Enter the name of the method to use for replotting the chromatogram and plotting the baseline.

You can specify a method with a different set of plotting parameters (attenuation, chart speed, zero), which will make the replotted chromatogram appear quite different from the original chromatogram. For example, one method can be optimized for the analysis and the other optimized for the replot and baseline.

When the replotting method contains timetable events, only integration functions 7, 8, 11, and 12 will affect the replot. Integration event 13 is forced on during the replot to display the result of functions 11 or 12 when they are used.

```
PROCESSING
```

(See page 8-17 for the replotted chromatogram.)

Specifying the Plotting Parameters Explicitly

```
Specify a method file [Y*/N]: N
```

10b. Enter N to indicate you want to enter the plotting parameters independently.

When you do not specify a method file:

- The parameters in the current method are ignored during the replot and the program prompts you for plotting parameters.
- The plotting parameters are set for the entire run and cannot be time-programmed.
- The processed peak data is plotted using the Baseline program plotting parameters to produce the chromatogram and baseline.
- No integrator functions are printed in the baseline replot.

```
Attenuation [0]:
```

11b. Press [ENTER] to accept the default or enter an attenuation value for the replot.

```
Chart speed [1.0]:
```

12. Press [ENTER] to accept the default or enter a chart speed value for the replot.

```
[ZERO] Position [0]:
```

13. Press [ENTER] to accept the default or enter a zero value for the replot.

```
Retention time labelling [Y*/N]:
```

14. Press [ENTER] to accept the current selection, Y.

```
Start/stop tick marks [Y*/N]:
```

15. Press [ENTER] to accept the current selection, Y.

```
Skip paper perforations in plot [Y/N*]:
```

16a. Press [ENTER] to accept the current selection, N.

If you choose to skip perforations during the replot, the program assumes that you have the top-of-form properly set. When you have supplied all of the program's parameters, the specified chromatogram is replotted and its baseline is drawn.

```
PROCESSING
```

(See page 8-17 for the replotted chromatogram.)

Replotting a Raw Signal File as Filtered Data

When the original chromatogram is source (raw) data, you can replot the chromatogram either from source (raw) or filtered data.

```
Source or filtered plot [S/F*]: [ENTER]
```

16b. Press [ENTER] to accept the current selection, F, for the replot.

In this example, raw data was stored for the original chromatogram. This gives you the choice of plotting source (raw) or filtered (analyzed) data.

```
PK WD [0.04]:
```

17b. Press [ENTER] to accept the current peak width value for the replot.

Since you choose to plot filtered data, you must supply a peak width value for the replotted chromatogram. The peak width entered here is used by the Baseline program to reanalyze the raw signal file and produce a bunched file (.BNA) that can be plotted.

```
Store bunched signal file on device [M:*]:
```

18. Press [ENTER] to accept the current default storage device M.

The bunched signal file (BNA) created during the reanalysis is temporarily stored on the storage device you specify; M, etc. It is deleted when the baseline is completed.

```
Skip paper perforations in plot [Y/N*]:
```

19. Press [ENTER] to accept the current selection, N.

If you choose to skip perforations during the replot, the program assumes that you have the top-of-form properly set. When you have supplied all of the program's parameters, the specified chromatogram is replotted and its baseline is drawn.

```
PROCESSING
```

(See page 8-17 for the replotted chromatogram.)

Replotting a Raw Signal File as Source Data

```
Source or filtered plot [S/F*]: S
```

16c. Enter S to select source plot type for the replot.

In this example, raw data was stored for the original chromatogram. This gives you the choice of plotting source (raw) or filtered (analyzed) data. If you store a bunched signal file for the original chromatogram, however, you could not obtain a source plot.

```
Skip paper perforations in plot [Y/N*]:
```

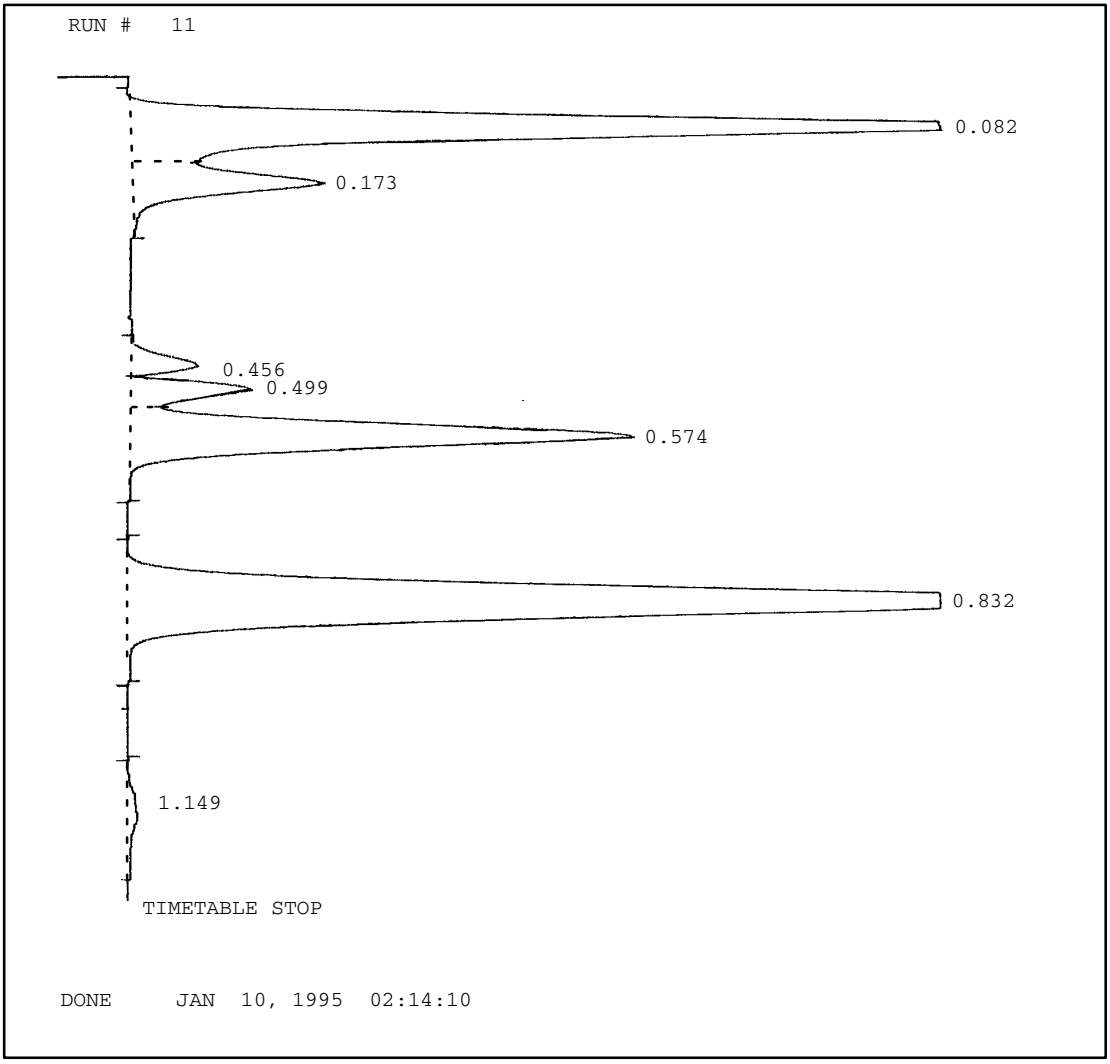
17c. Press [ENTER] to accept the current selection, N.

If you choose to skip perforations during the replot, the program assumes that you have the top-of-form properly set. When you have supplied all of the program's parameters, the specified chromatogram is replotted and its baseline is drawn.

```
PROCESSING
```

(See page 8-17 for the replotted chromatogram.)

The Replotted Chromatogram and Baseline



The Replotted Chromatogram with the Baseline Drawn.

When you specify a method for the plotting parameters, the replotted chromatogram has the following characteristics:

- The end of a solvent peak is marked with an S.
- Chart annotations for ZE, AT, CS, AR, PW, TH, IF, and ^ZE will appear on the replot as determined by the method.
- Only integration functions 7, 8, 11, and 12 of the method will affect the replot. Integration event 13 is forced on during the replot to display the result of functions 11 or 12 when they are used.

Examples

Plotting with the Current Method and Signal Files

```
WELCOME TO THE HP 3396 CHROMATOGRAM REPLOTTING PROGRAM (Rev. A.00.13)

At any prompt:      'Q'[ENTER] QUITS
                    'S'[ENTER] STARTS OVER


Use the current method and data files for the replot [Y*/N]: Y [ENTER]

PROCESSING
```

Replotting a Raw Signal File as Source Data

```
WELCOME TO THE HP 3396 CHROMATOGRAM REPLOTTING PROGRAM (Rev. A.00.13)

At any prompt:      'Q'[ENTER] QUITS
                    'S'[ENTER] STARTS OVER


Use the current method and data files for the replot [Y*/N]: N [ENTER]

Signal file for the replot [M:TESTA001.BNC]: M:RUN12.BNC [ENTER]

Processed peak file for the replot [M:TESTA001.PRO]: M:Q7BCB1C1.PRO [ENTER]

Start time for the replotted chromatogram [0.0]: [ENTER]

Stop time for the replotted chromatogram [3.27]: [ENTER]

Use the current method for plotting parameters [Y*/N]: [ENTER]

Specify a method file [Y*/N]: N [ENTER]

Attenuation [0]: 2 [ENTER]

Chart speed [1.0]: 2 [ENTER]

[ZERO] Position [0]: [ENTER]

Retention time labelling [Y*/N]: [ENTER]

Start/stop tick marks [Y*/N]: [ENTER]

Source or filtered plot [S/F*]: S [ENTER]

Skip paper perforations in plot [Y/N*]: [ENTER]

PROCESSING
```

Specifying the Plotting Parameters Explicitly

```
WELCOME TO THE HP 3396 CHROMATOGRAM REPLOTTING PROGRAM (Rev. A.00.13)

At any prompt:      'Q'[ENTER] QUITS
                    'S'[ENTER] STARTS OVER


Use the current method and data files for the replot [Y*/N]: N [ENTER]

Signal file for the replot [M:TESTA001.BNC]: M:RUN12.BNC [ENTER]

Processed peak file for the replot [M:TESTA001.PRO]: M:Q7BCB1C1.PRO [ENTER]

Start time for the replotted chromatogram [0.0]: [ENTER]

Stop time for the replotted chromatogram [3.27]: [ENTER]

Use the current method for plotting parameters [Y*/N]: [ENTER]

Specify a method file [Y*/N]: N [ENTER]

Attenuation [0]: 2 [ENTER]

Chart speed [1.0]: 2 [ENTER]

[ZERO] Position [0]: [ENTER]

Retention time labelling [Y*/N]: [ENTER]

Start/stop tick marks [Y*/N]: [ENTER]

Skip paper perforations in plot [Y/N*]: [ENTER]

PROCESSING
```

Specifying a Method for the Plotting Parameters

```
WELCOME TO THE HP 3396 CHROMATOGRAM REPLOTTING PROGRAM (Rev. A.00.13)

At any prompt:      'Q'[ENTER] QUITS
                    'S'[ENTER] STARTS OVER


Use the current method and data files for the replot [Y*/N]: N [ENTER]

Signal file for the replot [M:TESTA001.BNC]: M:RUN12.BNC [ENTER]

Processed peak file for the replot [M:TESTA001.PRO]: M:Q7BCB1C1.PRO [ENTER]

Start time for the replotted chromatogram [0.0]: [ENTER]

Stop time for the replotted chromatogram [3.27]: [ENTER]

Use the current method for plotting parameters [Y*/N]: N [ENTER]

Specify a method file [Y*/N]: [ENTER]

Method file [M:CURRENT.MET]: M:OLDMETH.MET [ENTER]

PROCESSING
```

Replotting a Raw Signal File as Filtered Data

```
WELCOME TO THE HP 3396 CHROMATOGRAM REPLOTTING PROGRAM (Rev. A.00.13)

At any prompt:      'Q'[ENTER] QUITS
                    'S'[ENTER] STARTS OVER

Use the current method and data files for the replot [Y*/N]: N [ENTER]
Signal file for the replot [M:TESTA001.BNC]: M:RUN12.BNC [ENTER]
Processed peak file for the replot [M:TESTA001.PRO]: M:Q7BCB1C1.PRO [ENTER]
Start time for the replotted chromatogram [0.0]: [ENTER]
Stop time for the replotted chromatogram [3.27]: [ENTER]
Use the current method for plotting parameters [Y*/N]: [ENTER]
Specify a method file [Y*/N]: N [ENTER]
Attenuation [0]: 2 [ENTER]
Chart speed [1.0]: 2 [ENTER]
[ZERO] Position [0]: [ENTER]
Retention time labelling [Y*/N]: [ENTER]
Start/stop tick marks [Y*/N]: [ENTER]
Source or filtered plot [S/F*]: [ENTER]
PK WD [0.04] .02 [ENTER]
Store bunched signal file on device [M:*]: [ENTER]
Skip paper perforations in plot [Y/N*]: [ENTER]
PROCESSING
```

Specifying the Signal Files to Plot

```
WELCOME TO THE HP 3396 CHROMATOGRAM REPLOTTING PROGRAM (Rev. A.00.13)

At any prompt:      'Q'[ENTER] QUITS
                    'S'[ENTER] STARTS OVER

Use the current method and data files for the replot [Y*/N]: N [ENTER]
Signal file for the replot [M:TESTA001.BNC]: M:RUN12.BNC [ENTER]
Processed peak file for the replot [M:TESTA001.PRO]: M:Q7BCB1C1.PRO [ENTER]
Start time for the replotted chromatogram [0.0]: [ENTER]
Stop time for the replotted chromatogram [3.27]: [ENTER]
Use the current method for plotting parameters [Y*/N]: [ENTER]
PROCESSING
```

Scheduling Baseline as an Autocall Program

You can automate the Baseline program by scheduling it as a postrun Autocall program. It will be loaded and run automatically at the end of each analysis.

Only one Autocall program can be assigned to function key **[0]** at a time.

* **ASSIGN 0, E:BASELINE.BAS**

1. **Assign E:BASELINE.BAS to function key [0] to schedule Baseline as an Autocall program.**

   After each run that follows, the chromatogram will be replotted and the baseline will be drawn using the current method, signal, and processed peak files.

2. **Press the integrator [START] button to start the run or sequence.**

   Do not press the key labelled **Auto Start** to start the run.

* **ASSIGN 0, E:AUTO_TOP.BAS**

3. **After the run or sequence is finished, return the program assignment of function key [0] to its original program, E:AUTO_TOP.BAS**

   If you fail to do this, when you try to use the Autoscheduler it will not operate correctly.

   The program called `AUTO_TOP` performs a number of prerun tests to check program space to ensure that the Autoscheduler can execute the postrun programs.