

HP Operations Agent

Software Version: 11.16

Windows®, HP-UX, Linux, Solaris, and AIX operating systems

User Guide

Document Release Date: March 2017
Software Release Date: March 2017



Legal Notices

Warranty

The only warranties for HP products and services are set forth in the express warranty statements accompanying such products and services. Nothing herein should be construed as constituting an additional warranty. HP shall not be liable for technical or editorial errors or omissions contained herein.

The information contained herein is subject to change without notice.

Restricted Rights Legend

Confidential computer software. Valid license from HP required for possession, use or copying. Consistent with FAR 12.211 and 12.212, Commercial Computer Software, Computer Software Documentation, and Technical Data for Commercial Items are licensed to the U.S. Government under vendor's standard commercial license.

Copyright Notice

© Copyright 2012-2017 Hewlett-Packard Development Company, L.P.

Trademark Notices

Adobe ® is a trademark of Adobe Systems Incorporated.

Microsoft® and Windows® are U.S. registered trademarks of the Microsoft group of companies.

UNIX® is a registered trademark of The Open Group.

This product includes an interface of the 'zlib' general purpose compression library, which is Copyright © 1995-2002 Jean-loup Gailly and Mark Adler.

Acknowledgements

This product includes cryptographic software written by Eric Young (eay@cryptsoft.com).

This product includes software developed by the OpenSSL Project for use in the OpenSSL Toolkit (<http://www.openssl.org/>).

This product includes software written by Tim Hudson (tjh@cryptsoft.com).

This product includes an interface of the 'zlib' general purpose compression library, which is Copyright ©1995-2002 Jean-loup Gailly and Mark Adler.

Documentation Updates

The title page of this document contains the following identifying information:

- Software Version number, which indicates the software version.
- Document Release Date, which changes each time the document is updated.
- Software Release Date, which indicates the release date of this version of the software.

To check for recent updates or to verify that you are using the most recent edition of a document, go to:
<https://softwaresupport.hp.com>

This site requires that you register for an HP Passport and sign in. To register for an HP Passport ID, go to:
<https://hpp12.passport.hp.com/hppcf/createuser.do>

Or click the **the Register** link at the top of the HP Software Support page.

You will also receive updated or new editions if you subscribe to the appropriate product support service. Contact your HP sales representative for details.

Support

Visit the HP Software Support Online web site at: <https://softwaresupport.hp.com>

This web site provides contact information and details about the products, services, and support that HP Software offers.

HP Software online support provides customer self-solve capabilities. It provides a fast and efficient way to access interactive technical support tools needed to manage your business. As a valued support customer, you can benefit by using the support web site to:

- Search for knowledge documents of interest
- Submit and track support cases and enhancement requests
- Download software patches
- Manage support contracts
- Look up HP support contacts
- Review information about available services
- Enter into discussions with other software customers
- Research and register for software training

Most of the support areas require that you register as an HP Passport user and sign in. Many also require a support contract. To register for an HP Passport ID, go to:

<https://hpp12.passport.hp.com/hppcf/createuser.do>

To find more information about access levels, go to:

<https://softwaresupport.hp.com/web/softwaresupport/access-levels>

HP Software Solutions Now accesses the HPSW Solution and Integration Portal Web site. This site enables you to explore HP Product Solutions to meet your business needs, includes a full list of Integrations between HP Products, as well as a listing of ITIL Processes. The URL for this Web site is

<http://h20230.www2.hp.com/sc/solutions/index.jsp>

Contents

Chapter 1: Introduction	19
Documentation Map	19
Chapter 2: Managing Data Collection	21
Using the Log File-Based Data Store	21
Collection Log Files	21
logglob	23
logappl	23
logproc	23
logpcmd	23
logdev	24
logtran	24
logls	24
logindx	24
Scope Status	24
parm File	24
Modify the parm File	26
parm File Parameters	27
Parameter Descriptions	31
ID	32
Log	32
Thresholds	33
Procthreshold	34
appthreshold	35
diskthreshold	35
bynetifthreshold	35
fsthreshold	35
lvthreshold	35
bycputhreshold	36
scopetransactions	36
subprocinterval	36
gapapp	36
fstypes	37
wait	38
Size	38
Mainttime	39
Days	39
Maintweekday	39
javaarg	40

Flush	40
project_app	40
proclist	41
approc	41
proccmd	41
ignore_mt	41
cachemem	45
kvm_collection_mode	46
Application Definition Parameters	46
Application	47
File	47
argv1	49
cmd	49
User	49
Group	50
Or	50
Priority	51
Application Definition Examples	51
Configure Data Logging Intervals	52
Configuring Data Collection on vMA Nodes	52
Configuring Data Collection for Frames	52
Task 1: Configure Passwordless SSH Access	53
Task 2: Enable Frame Utilization Monitoring on the HMC System	53
Task 3: Configure the HP Operations Agent	53
Enabling the Global and Process System Call Metrics for GlancePlus on Linux	54
Configuring the Metric Collection using init_fttrace.sh	54
Configuring the Metric Collection using manual steps	55
Collection of BYCORE and THREAD SYSCALL metric values in advisory mode on Linux	57
Normalizing CPU Metrics on Hyper-Threading/Simultaneous Multi-Threading-Enabled Systems	58
Logging Metrics Calculated with the Core-Based Normalization	58
Stopping and Restarting Data Collection	59
Stopping Data Collection	60
Restarting Data Collection	60
Daylight Savings	61
Changing System Time Manually	61
Effective Data Collection Management	61
Controlling Disk Space Used by Log Files	61
Setting mainttime	62
Setting the Maximum Log File Size	62
Managing Your Resizing Processes	63
Data Archiving	63
Managing Your Archiving Processes	64

Chapter 3: Working with the HP Operations Agent	65
Configuring the Monitor Agent	65
Configure the Agent to Monitor MIB Objects	65
Persistence of Monitored Object	66
Configuring the Event Interceptor	67
Message Storm	69
Configure Message Storm Detection Capabilities	70
Checking the Message Rate	73
Integrating HP Operations Agent with HP NNMi	73
Integrating with NNMi Northbound Interface to Associate the Message Severity	74
Configure SNMP Trap Interceptor to Enhance Message Severity	74
Integrating NNMi Northbound Interface to Add CMA on the HPOM Console	75
Configure SNMP Trap Interceptor to Create CMAs from NNMi CIA on the HPOM Console	76
Integrating with NNMi Trap Forwarding Interface to Assign Source of SNMP Trap	77
Configure SNMP Trap Interceptor to Derive the Source Node Name	78
Configuring the Backup Server	78
Configuring the RTMA Component	80
Checking the License for Perfd Process	80
Modifying the Settings	80
Monitoring HP Operations agent on IPv6 connections	82
Restrict Access	82
Configuring the Agent User	83
Requirements for Using a Non-Default User	83
Limitations of Using a Non-Default User	84
Configure the Agent User During Installation	85
Configure the Agent User After Installation	88
Change the Default User on Windows	88
Alternative Method: Use the ovswitchuser Command	88
Change the Default User on UNIX/Linux	90
Use a Profile File	90
Alternative Method: Use the ovswitchuser Command	91
Change the Default User for Commands	93
Configuring the Security Component Variables	93
Configuring the Security Component for Symmetric Key	95
Configuring viserver for Monitoring vMA Nodes	96
viserver.properties	97
port	97
hosts	97
instance	97
jvmArgs	97
log4jInterval	98
VILog4j.xml	98

Monitoring Applications and Services Logs on Windows	99
Monitor Applications and Services Event Logs from HPOM for Windows	100
Monitor Applications and Services Event Logs from HPOM on UNIX/Linux 9.1x	101
Monitor Applications and Services Event Logs from HPOM for UNIX 8.35	102
 Chapter 4: Overview of Node Resolution	 103
 Chapter 5: Using the Utility Program	 108
Running the Utility Program	108
Using Interactive Mode	109
Example of Using Interactive and Batch Mode	109
Utility Command Line Interface	110
Example of Using the Command Line Interface	111
Utility Scan Report Details	112
Scan Report Information	113
Initial Values	113
Initial Parm File Global Information	113
Initial Parm File Application Definitions	114
Chronological Detail	114
Parm File Global Change Notifications	114
Parm File Application Addition/Deletion Notifications	115
scope Off-Time Notifications	115
Application-Specific Summary Report	115
Summaries	116
Process Log Reason Summary	116
Scan Start and Stop	117
Application Overall Summary	117
Collector Coverage Summary	118
Log File Contents Summary	118
Log File Empty Space Summary	119
 Chapter 6: Utility Commands	 120
analyze	121
checkdef	122
detail	123
exit	124
guide	124
help	124
list	125
logfile	126

menu	127
parmfile	128
quit	129
resize	129
Resize Command Reports	131
Examples	132
scan	133
sh	134
show	135
start	136
stop	137
Chapter 7: Using the Extract Program	139
Running the Extract Program	140
Syntax	140
Using Interactive Mode	140
Extract Command Line Interface	141
Overview of the Export Function	144
How to Export Data	144
Sample Export Tasks	146
Generating a Printable CPU Report	146
Producing a Customized Export File	146
Export Data Files	146
Export Template File Syntax	147
Parameters	148
Export File Title	150
Creating a Custom Graph or Report	150
Output of Exported Files	151
Notes on ASCII and Datafile Formats	152
Notes on Binary Format	153
Binary Header Record Layout	153
Binary Title Record	153
Binary Item Identification Record	154
Binary Scale Factor Record	154
Special Scale Factors	154
Special Scale Factors	155
Transaction Name Record	155
Disk Device Name Record	155
Logical Volume Name Record	155
Netif Name Record	156

Chapter 8: Extract Commands	157
application	161
class	162
configuration	163
cpu	164
disk	165
exit	166
export	166
extract	168
filesystem	170
global	171
guide	172
help	172
list	173
logfile	174
lvolume	175
menu	176
monthly	178
netif	180
output	181
process	183
quit	184
report	184
sh	184
shift	185
show	186
start	188
stop	189
transaction	191
weekdays	192
weekly	193
yearly	194
Chapter 9: Using the cpsd Program	197
Using the Interactive Mode	197

View Real-Time Metrics	198
Modify a Metric Class	198
View All the Available Metrics	198
Organize a Metric Class	198
View Metric Help	199
View Summarized Metric Data	199
Enabling Threshold and Filters Conditions	199
Chapter 10: Adviser for the RTMA Component	201
Alarms and Symptoms	201
Working of the Adviser Script	201
Using Adviser	202
Run the Adviser Script on Multiple Systems	202
Adviser Syntax	203
Syntax Conventions	203
Comments	203
Conditions	203
Constants	204
Expressions	204
Metric Names in Adviser Syntax	205
Printlist	206
Variables	206
ALARM Statement	206
ALERT Statement	207
ALIAS Statement	207
ASSIGNMENT Statement	208
COMPOUND Statement	208
EXEC Statement	208
IF Statement	209
LOOP Statement	209
PRINT Statement	210
SYMPTOM Statement	210
Chapter 11: Performance Alarms	212
Processing Alarms	212
Alarm Generator	212
Sending SNMP Traps to Network Node Manager	213
Sending Messages to HPOM	213
Executing Local Actions	213
Errors in Processing Alarms	214
Analyzing Historical Data for Alarms	215
Examples of Alarm Information in Historical Data	215
Alarm Definition Components	216

Alarm Syntax Reference	216
Alarm Syntax	216
Conventions	217
Common Elements	217
Comments	218
Compound Statements	218
Conditions	218
Constants	219
Expressions	219
Metric Names	219
Messages	220
ALARM Statement	221
Syntax	221
How It Is Used	222
Examples	223
ALERT Statement	224
Syntax	224
How It Is Used	225
Example	225
EXEC Statement	225
Syntax	225
How It Is Used	226
Examples	226
PRINT Statement	227
Syntax	227
Example	227
IF Statement	227
Syntax	227
How It Is Used	228
Example	228
LOOP Statement	229
Syntax	229
How It Is Used	229
Example	229
INCLUDE Statement	230
Syntax	230
How It Is Used	230
Example	230
USE Statement	230
Syntax	231
How It Is Used	231
VAR Statement	232
Syntax	232
How It Is Used	232
Examples	233
ALIAS Statement	233

Syntax	233
How It Is Used	233
Examples	233
SYMPTOM Statement	234
Syntax	234
How It Is Used	234
Example	235
Alarm Definition Examples	235
Example of a CPU Problem	235
Example of Swap Utilization	235
Example of Time-Based Alarms	236
Example of Disk Instance Alarms	236
Customizing Alarm Definitions	237
Chapter 12: Using the Performance Collection Component on Windows	238
Data Types and Classes	239
Summarization Levels	239
Ranges of Data to Extract or Export	240
Extracting Log File Data	240
Exporting Log File Data	241
File Attributes	242
File Format	242
Missing Value	243
Field Separators	243
Summary Minutes	243
Headings	243
Multiple Layout	243
Export File Title	243
Export File Templates	244
Default Export Files	244
scopent Data	245
DSI Data	246
Making a Quick Export Template	246
Selecting Metrics to Export	247
Saving Your Selections	247
Configuring Export Templates	247
Selecting Metrics for Export	248
Saving Your Selections	249
Archiving Log File Data	249
Archival Periods	250
Appending Archived Data	250
Archiving Tips	251

Analyzing a Log File	251
Range of Data to be Analyzed	252
Analysis Report	252
Scanning a Log File	253
Resizing a Log File	254
Configuring User Options	256
Configuring Collection Parameters	257
Configuring Alarm Definitions	258
Configuring Data Sources	259
Data Sources File Format	259
Configuring Data Sources from Remote Locations	260
Configuring Transactions	261
Configuring Persistent DSI Collections	263
Checking Performance Collection Component Status	264
Running Processes	264
Datacomm Services	265
System Services	265
System Configuration	265
File Version Numbers	265
Status File Latest Entries	265
Status File Warnings and Errors	265
Building Collections of Performance Counters	266
Building a Performance Counter Collection	266
Managing a Performance Counter Collection	266
Tips for Using Extended Collection Builder and Manager	266
Administering ECBM from the Command line	267
Chapter 13: Overview of Data Source Integration	269
How DSI Works	269
Creating the Class Specification	270
Collecting and Logging the Data	270
Using the Data	270
Chapter 14: Using Data Source Integration	271
Planning Data Collection	271
Defining the Log File Format	271
How Log Files Are Organized	272
Creating the Log File Set	273
Testing the Class Specification File and the Logging Process (Optional)	274
Logging Data to the Log File Set	274

Using the Logged Data	275
Chapter 15: DSI Class Specification Reference	276
Class Specifications	276
Class Specification Syntax	277
CLASS Description	277
Syntax	278
Default Settings	278
CLASS	278
LABEL	279
INDEX BY, MAX INDEXES, AND ROLL BY	279
Controlling Log File Size	282
RECORDS PER HOUR	284
CAPACITY	284
Managing Data with sdlutil	285
METRICS	285
LABEL	286
Summarization Method	287
PRECISION	288
TYPE TEXT LENGTH	288
Sample Class Specification	289
Chapter 16: DSI Program Reference	292
sdlcomp Compiler	292
Compiler Syntax	292
Sample Compiler Output	293
Configuration Files	296
Defining Alarms for DSI Metrics	296
Alarm Processing	297
dsilog Logging Process	297
Syntax	298
How dsilog Processes Data	299
Testing the Logging Process with Sdlogdata	299
Creating a Format File	302
Changing a Class Specification	303
Exporting DSI Data	303
Example of Using Extract to Export DSI Log File Data	304
Viewing Data in Performance Manager	304
Managing Data with sdlutil	304
Syntax	304

Chapter 17: Examples of Data Source Integration	306
Writing a dsilog Script	306
Example 1 - Problematic dsilog Script	306
Example 2 - Recommended dsilog Script	307
Logging vmstat Data	307
Creating a Class Specification File	307
Compiling the Class Specification File	308
Starting the dsilog Logging Process	309
Accessing the Data	309
Logging sar Data from One File	310
Creating a Class Specification File	310
Compiling the Class Specification File	312
Starting the DSI Logging Process	313
Logging sar Data from Several Files	314
Creating Class Specification Files	315
Compiling the Class Specification Files	322
Starting the DSI Logging Process	322
Logging sar Data for Several Options	323
Logging the Number of System Users	332
Chapter 18: Error Message	334
SDL Error Messages	334
DSILOG Error Messages	334
General Error Messages	334
Chapter 19: What is Transaction Tracking?	336
Improving Performance Management	336
Benefits of Transaction Tracking	336
Client View of Transaction Times	336
Transaction Data	337
Service Level Objectives	337
A Scenario: Real Time Order Processing	337
Requirements for Real Time Order Processing	338
Preparing the Order Processing Application	338
Monitoring Transaction Data	338
Guidelines for Using ARM	339
Chapter 20: How Transaction Tracking Works	341
Support of ARM 2.0	341

Support of ARM API Calls	341
arm_complete_transaction Call	342
Sample ARM-Instrumented Applications	342
Specifying Application and Transaction Names	343
Transaction Tracking Daemon (ttd)	343
ARM API Call Status Returns	344
Measurement Interface Daemon (midaemon)	345
Transaction Configuration File (ttd.conf)	345
Adding New Applications	345
Adding New Transactions	346
Changing the Range or SLO Values	346
Configuration File Keywords	346
tran	347
range	347
slo	347
Configuration File Format	348
Configuration File Examples	349
Overhead Considerations for Using ARM	350
Guidelines	350
Disk I/O Overhead	351
CPU Overhead	351
Memory Overhead	351
Chapter 21: Getting Started with Transactions	353
Before you start	353
Setting Up Transaction Tracking	353
Defining Service Level Objectives	354
Modifying the Parm File	354
Collecting Transaction Data	355
Error Handling	355
Limits on Unique Transactions	355
Customizing the Configuration File (optional)	356
Monitoring Performance Data	357
Alarms	358
Chapter 22: Transaction Tracking Messages	359
Chapter 23: Transaction Metrics	360
Chapter 24: Transaction Tracking Examples	361
Pseudocode for Real Time Order Processing	361

Configuration File Examples	364
Example 1 (for Order Processing Pseudocode Example)	364
Example 2	364
Example 3	365
Example 4	365
Chapter 25: Advanced Features	366
How Data Types Are Used	366
User-Defined Metrics	367
scope Instrumentation	368
Chapter 26: Transaction Libraries	369
ARM Library (libarm)	369
C Compiler Option Examples by Platform	372
ARM NOP Library	373
Using the Java Wrappers	374
Examples	374
Setting Up an Application (arm_init)	374
Syntax:	374
Setting Up a Transaction (arm_getid)	374
Setting Up a Transaction With UDMs	374
Adding the Metrics	375
Setting the Metric Data	375
Setting Up a Transaction Without UDMs	376
Setting Up a Transaction Instance	376
Starting a Transaction Instance (arm_start)	376
Starting the Transaction Instance Using Correlators	376
Requesting a Correlator	376
Passing the Parent Correlator	377
Requesting and Passing the Parent Correlator	377
Retrieving the Correlator Information	377
Starting the Transaction Instance Without Using Correlators	377
Updating Transaction Instance Data	378
Updating Transaction Instance Data With UDMs	378
Updating Transaction Instance Data Without UDMs	378
Providing a Larger Opaque Application Private Buffer	378
Stopping the Transaction Instance (arm_stop)	379
Stopping the Transaction Instance With a Metric Update	379
Stopping the Transaction Instance Without a Metric Update	379
Using Complete Transaction	379

Using Complete Transaction With UDMs:	380
Using Complete Transaction Without UDMs:	380
Further Documentation	380
Chapter 27: Logging and Tracing	381
Logging	381
Configure the Logging Policy	382
Tracing	382
Identify the Application	382
Set the Tracing Type	384
Introduction to the Trace Configuration File	385
Syntax	385
Create the Configuration File	385
Enabling Tracing and Viewing Trace Messages with the Command-Line Tools	385
Enabling Tracing and Viewing Trace Messages with the Tracing GUI	386
Enable the Tracing Mechanism	386
View Trace Messages	387
Use the Trace List View	389
Use the Procedure Tree View	390
Filter Traces	390
Using the Tracing GUI	392
Chapter 28: Troubleshooting	398
Operations Monitoring Component	398
Performance Collection Component	402
RTMA	404
HP GlancePlus	405
Send Documentation Feedback	406

Chapter 1: Introduction

The HP Operations agent introduces many services, processes, and utilities on the system. Command-line utilities help you configure the operation and monitor the performance of the agent. Using certain command-line utilities, you can view the real-time system performance data captured by the agent. Utilities like the tracing tools help you view the diagnostic information of the agent for troubleshooting.

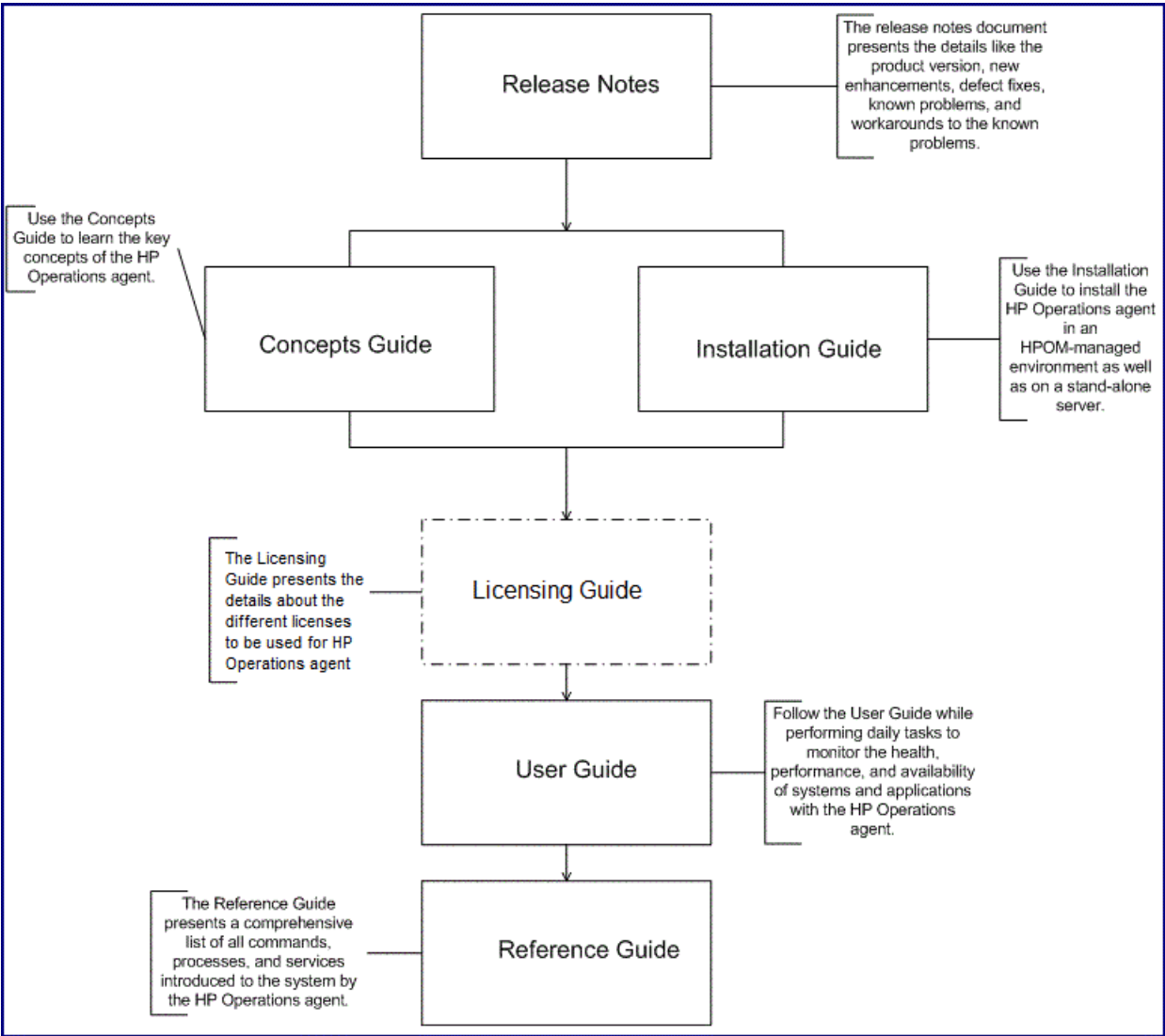
The HP Operations agent offers you a series of configuration variables; these variables help you control the behavior of the agent. You can use the `ovconfchg` command to assigned desired values to these variables.

This guide contains information on the command-line utilities, services, and processes introduced on the system by the HP Operations agent. The guide also provides you with a list of configuration variables that you can use while configuring the default behavior of the HP Operations agent.

Documentation Map

The documentation map presents a list of all the major documents for the HP Operations agent. You can use the map to identify the necessary document when you need assistance.

Figure 1: Documentation Map for the HP Operations Agent



Chapter 2: Managing Data Collection

The HP Operations agent provides you with a data collector to collect and log system performance data of the monitored system. The data collector program—**scope**—enables you to store the collected data on the system. You can view and analyze the stored data using HP Performance Manager or HP Reporter.

The scope collector enables you to perform the following tasks on the system:

- Gather metric data that indicates health and performance of the monitored system
- Log the collected metric data into different log files

You can view the data logged into these log files with the help of the following tools:

- The extract and utility commands
- Data analysis programs (such as HP Performance Manager)

The configuration parameter file—the **parm** file—enables you to configure the default data logging mechanism of the scope collector. By modifying parameters in the **parm** file, you can control the following properties of the scope collector:

- Data logging interval
- Types of data
- Size of log files

After installing the HP Operations agent on the node, you must configure the data collection mechanism of scope by modifying the **parm** file.

Using the Log File-Based Data Store

Older versions of the HP Operations agent (older than the version 11.00) used to store data into the embedded performance component (EPC). The HP Operations agent 11.00 (or higher) stores the system performance data into the log file-based data store. However, the EPC data store is still available for different Smart Plug-ins (SPIs).

When a SPI collector detects the presence of both the EPC and log file-based data store, the data collected by the SPI automatically gets logged into the log file-based data store.

Collection Log Files

The scope data collector (scopeux on UNIX and Linux nodes; scopent on Windows nodes) collects and summarizes performance measurements of system-resource utilization and records the data into the following log files, depending on the data classes specified in the log line of the **parm** file:

[logglob](#)

[logappl](#)

logproc

logpcmd

logdev

logtran

logls

logindx

Note: The time-stamps of the records in the log files indicate the starting time of data collection. The concept of interesting processes is a filter that helps minimize the volume of data logged and is controlled from the parm file. Scope does not log NFS data but you can view the NFS data through GlancePlus on the local file system. by default. You can configure the fstypes parameter to log NFS data (see "[fstypes](#)" on page 37).

When you roll back to an older version of the agent, the scope collector may stop logging data into the log files. To resolve this, follow these steps:

Log on to the node as an administrator or root.

Stop the scope collector:

On Windows:

```
%ovinstalldir%bin\ovpacmd stop col
```

On HP-UX, Linux, and Solaris:

```
/opt/perf/bin/ovpa stop scope
```

On AIX

```
/usr/lpp/perf/bin/ovpa stop scope
```

Go to the following directory command:

On Windows:

```
%ovdatadir%datafiles
```

On UNIX/Linux:

```
/var/opt/perf/datafiles
```

Take a backup of all log files for future use, and then delete all the files in the directory.

You can transfer the backed-up log files on a system where the version 11.16 is running and use the extract, export, or utility command or a data analysis tool (such as HP Performance Manager) to view the data that was collected by the version 11.16 of the HP Operations agent.

After you take a backup of log files, delete all files under the **datafiles** directory.

Start the scope collector:

On Windows:

```
%ovinstalldir%bin\ovpacmd start col
```

On HP-UX, Linux, or Solaris:

```
/opt/perf/bin/ovpa start scope
```

On AIX:

```
/usr/lpp/perf/bin/ovpa start scope
```

logglob

The **logglob** file contains measurements of system-wide (global) resource utilization information. The scope collector summarizes the global data and periodically records the data at intervals specified in the **parm** file.

logappl

The **logappl** file contains aggregate measurements of processes that run in applications defined in the **parm** file. The scope collector summarizes the application data and periodically records the data at intervals specified in the **parm** file.

logproc

The scope collector identifies processes that might be of your interest, and then records the aggregate measurements of identified processes in the **logproc** file. Scope identifies processes for based on the following conditions:

Beginning of a process

End of a process

Configuration details specified in the **parm** file

logpcmd

The **logpcmd** file contains the details of command-line activities performed on the processes that are logged into the **logproc** file.

Note: You cannot control the size, rollover, and logging interval for the **logpcmd** file.

The **logpcmd** file is stored into the following directory on the node:

On Windows: **%ovdatadir%\datafiles**

On UNIX (and Linux): **/var/opt/perf/datafiles**

The file can store a maximum of 25 MB of data. When the data collection mechanism starts, the scope collector creates the first instance of the **logpcmd** file with the extension 0. When the **logpcmd0** file reaches the 25 MB limit, scope creates the second instance of the **logpcmd** file—the **logpcmd1** file.

If the **logpcmd1** file exceeds the 25 MB limit, data starts to roll over from the **logpcmd0** file.

logdev

The **logdev** file contains measurements of individual device performance. The scope collector summarizes the device data and periodically records the data at intervals specified in the **parm** file.

logtran

The **logtran** file contains measurements of ARM transaction data. The scope collector summarizes the transaction data and periodically records the data at intervals specified in the **parm** file. For more information on collecting transaction data, see [What is Transaction Tracking?](#)

logls

The **logls** file contains information about the logical systems. The scope collector summarizes the logical system data and periodically records the data at intervals specified in the **parm** file.

The **logls** file is available only on the HP Operations agent for HPVM, Hyper-V Host, vSphere Management Assistant (vMA), Solaris global zones, KVM, Xen, and AIX-LPAR.

logindx

The **logindx** file contains information needed to access data in the other log files.

Scope Status

In addition to the log files, two other files are created when scope is started. They are the **RUN** file that resides in the **/var/opt/perf/datafiles/** directory and the **status.scope** file that resides in the **/var/opt/perf/** directory.

The **RUN** file is created to indicate that the scope process is running. Removing this file causes scope to terminate.

The **/var/opt/perf/status.scope** file serves as a status/error log for the scope process. New information is appended to this file each time the scope collector is started, stopped, or when a warning or error is encountered. To view the most recent status and error information from scope, use the `perfstat -t` command.

parm File

The **parm** file is a text file containing the instructions that tell scope to log specific performance measurements.

During fresh installation, the HP Operations agent places the default **parm** file into two different directories:

On Windows:

Note: On Windows, the parm file exists with the extension **.mwc (parm.mwc)**.

%ovinstalldir%\newconfig

%ovdatadir%

On HP-UX, Solaris, and Linux:

/opt/perf/newconfig

/var/opt/perf

On AIX:

/usr/lpp/perf/newconfig

/var/opt/perf

The data collection mechanism of scope is controlled by the settings in the **parm** file located into the %ovdatadir% (for Windows) or /var/opt/perf (for UNIX or Linux) directory.

If you want to modify the default collection mechanism, you must modify the settings in the **parm** file that is located into the %ovdatadir% (for Windows) or /var/opt/perf (for UNIX or Linux) directory.

When you upgrade the HP Operations agent on a node (from an older version of the HP Performance Agent), the upgrade process updates the copy of the **parm** file available in the **newconfig** directory. The **parm** file that resides into the other directory remains unaffected and continues to govern the data collection mechanism on the node. This method, in effect, enables you to retain the configured data collection mechanism even after upgrade of the product. You can, any time after the product upgrade, compare the existing configuration settings of the **parm** file with the new version of the **parm** file available in the **newconfig** directory, and then make necessary changes.

The **parm** file is set up to collect an average amount of log file data. The maximum amount depends on your system. See the description of the parameter size in [Parameter Descriptions](#).

On all UNIX/Linux systems, the m4 macro processor utility is used for preprocessing the **parm** file and there is only one **parm** file, irrespective of the different platforms. The Performance Collection Component preprocesses the generic **parm** file to create a run-time **parm** file which is a text file. You can use the following command to generate a run-time **parm** file:

```
# m4 -DPARMOS=<Operatingsystem>/var/opt/perf/parm > parm.m4
```

Set the appropriate value for <Operatingsystem>.

Note: The value of <Operatingsystem> is case sensitive. To get an appropriate value for the operating system, run the following command:

```
uname
```

For example:

If you run the `uname` command on a Linux operating system, the following output is generated:

Linux

The **parm** file is available at the location `/var/opt/perf/parm` for the new installations. The pre-existing **parm** files, without the m4 utility, work for the previous versions of HP Operations agent.

The system checks for m4 at the time of installation. If m4 is not present, make sure that you install m4 on the system. For more information, see the section *Prerequisites for Installing HP Operations agent* in the *HP Operations Agent and HP Operations Smart Plug-in for Infrastructure Installation Guide*.

You can add custom application definitions to the external file and include it to the **parm** file by using the command `#include(var/opt/perf/parm.apps)`. Refer the *Application* section of the **parm** file for more information.

Modify the parm File

You can modify the **parm** file using any word processor or editor that can save a file in the ASCII format.

When you modify the **parm** file, or create a new one, the following rules and conventions apply:

Any parameter you specify overrides the default values. See the **parm** file available in the **newconfig** directory for the default values.

The order in which the parameters are specified into the **parm** file is not important.

If you specify a parameter more than once, the last instance of the parameter takes effect.

The file, user, group, cmd, argv1, and or parameters must follow the application statement that they define.

Application parameters must be listed in order so that a process will be aggregated into the application when it is first matched.

You can use uppercase letters, lowercase letters, or a combination of both for all commands and parameter statements.

You can use blank spaces or commas to separate key words in each statement.

You can comment parameters in the **parm** file. Any line starting with a comment code (`/*`) or pound sign (`#`) is ignored.

After modifying the **parm** file, you must restart the Performance Collection Component for the changes to take effect. To restart the Performance Collection Component, run the following command:

On Windows:

```
%ovinstalldir%bin\ovpacmd REFRESH COL
```

On HP-UX, Linux, or Solaris:

```
/opt/perf/bin/ovpa -restart scope
```

On AIX:

```
/usr/lpp/perf/bin/ovpa -restart scope
```

If you want to use the Real-Time Metric Access (RTMA) component, you must also restart the perfcd process:

On Windows:

```
%ovinstalldir%bin\ovpacmd REFRESH RTMA
```

On HP-UX, Linux, or Solaris:

```
/opt/perf/bin/pctl restart
```

On AIX:

```
/usr/lpp/perf/bin/pctl restart
```

parm File Parameters

Scope is controlled by specific parameters in the collection parameters (**parm**) file that do the following:

- Set maximum amount of disk space for the raw scope log files.
- Specify data types to be logged.
- Specify the interval at which data should be logged.
- Specify attributes of processes and metrics to be logged.
- Define types of performance data to be collected and logged.
- Specify the user-definable sets of applications that should be monitored. An application can be one or more programs that are monitored as a group.
- Specify when scope should perform daily log file maintenance activities so that they do not impact system availability.

You can modify these parameters to configure scope to log performance data that match the requirements of the monitored system (see [Modify the parm File](#))

Note: Collection of system configuration parameters happen only when you start, add or remove devices from the system.

The **parm** file parameters listed in the table below are used by scope. Some of these parameters are for specific systems as indicated in the table. For detailed descriptions of these parameters, see [Parameter Descriptions](#) and [Application Definition Parameters](#).

parm File Parameters Used by scope

Parameter	Values or Options
id	system ID
log	global application [=prm] [=all] ([=prm] on HP-UX only) process device=disk,lvm,cpu,filesystem,all(lvm on HP-UX only,) transaction=correlator,resource (resource on HP-UX only) logicalsystem (For Solaris, logical system is supported on Solaris 10 operating

parm File Parameters Used by scope, continued

Parameter	Values or Options
	<p>environment or later)</p> <p>In AIX, logical system is supported on LPAR on AIX 5L V5.3 ML3 and later and WPAR on AIX 6.1 TL2 Global environment only.</p> <p>For enabling lpar logging, logicalsystems=lpar logicalsystems</p> <p>For enabling wpar logging, logicalsystems=wpar</p> <p>For enabling both lpar and wpar logging, logicalsystems=lpar,wpar logicalsystems=wpar,lpar logicalsystems=all</p>
mainttime	hh:mm (24-hour time format)
scopetransactions	onoff
subprocinterval	value in seconds (not on HP-UX)
javaarg	truefalse
	NOTE: Only on UNIX/Linux.
procthreshold (same as threshold)	<p>cpu=<i>percent</i></p> <p>disk=<i>rate</i> (not on Linux or Windows)</p> <p>memory=nn (values in MBs)</p> <p>nonew</p> <p>nokilled</p> <p>shortlived</p>
appthreshold	cpu= <i>percent</i>
diskthreshold	util= <i>rate</i>
bynetifthreshold	iorate= <i>rate</i>
fsthreshold	util= <i>rate</i>
lvthreshold	iorate= <i>rate</i>
bycputhreshold	cpu= <i>percent</i>

parm File Parameters Used by scope, continued

Parameter	Values or Options
fstypes	To include only specific file systems for data logging, use the syntax <code><file_system1>, <file_system2>, <file_system3>, ...</code> To exclude a file system, use the syntax <code>!<file_system></code> .
wait	<code>cpu=percent</code> (HP-UX only) <code>disk=percent</code> (HP-UX only) <code>mem=percent</code> (HP-UX only) <code>sem=percent</code> (HP-UX only) <code>lan=percent</code> (HP-UX only)
application	<i>application name</i>
file	<i>file name [, ...]</i>
argv1	first command argument [,]
cmd	command line regular expression
user	<i>user login name [,]</i>
group	<i>groupname [,]</i>
or	
priority	low value-high value (range varies by platform)
size	(values are in MBs) <code>process=nn</code> (the maximum value is 4096) The maximum value for all the below classes is 2048. <code>global=nn application=nn</code> <code>device=nn</code> <code>transaction=nn</code> <code>logicalsystem=nn</code>
days	<code>global=nn</code> (values are in days) <code>application=nn</code> <code>process=nn</code> <code>device=nn</code> <code>transaction=nn</code> <code>logicalsystem=nn</code>
maintweekday	Sun Mon Tue Wed Thu Fri Sat
collectioninterval	<code>process=ss</code> (values in seconds) <code>global=ss</code>

parm File Parameters Used by scope, continued

Parameter	Values or Options
gapapp	blank unassignedprocesses existingapplicationname other
Flush	ss(values in seconds) 0 (disables data flush)
project_app NOTE: Only on Solaris	true false (only on Solaris 10 and above)
proccmd NOTE: Only on UNIX/Linux.	0 (disables logging of process commands) nnnn (refers to the numeric value of the length of a process command. Maximum value is 1024)
proclist NOTE: Only on Solaris	all (the Performance Collection Component in the global zone monitors all the processes that are running in the global and non-global zones) local (the Performance Collection Component in the global zone monitors only the processes that are running in the global zone) This parameter has no effect in non-global zones.
appproc NOTE: Only on Solaris	all (configures the Performance Collection Component to calculate APP_ metrics with processes for applications that belong to the global and non-global zones) local (configures the Performance Collection Component to calculate APP_ metrics with processes for applications that belong to the global zone only) This parameter has no effect in non-global zones.
ignore_mt	true(CPU metrics of global class report values normalized against the active number of cores in the system) false(CPU metrics of global class report values normalized against active number of CPU threads in the system) ineffective(multithreading is turned off) NOTE: This parameter has no effect on HP-UX. You must run the midaemon -ignore_mt command on HP-UX to switch between the above modes. For more information, see Logging Metrics Calculated with the Core-Based Normalization .
cachemem	f or free (The HP Operations agent does not include the buffer cache size)

parm File Parameters Used by scope, continued

Parameter	Values or Options
	when calculating the value of the GBL_MEM_UTIL metric) u or user (The HP Operations agent includes the buffer cache size when calculating the value of the GBL_MEM_UTIL metric)
kvm_collection_ mode	session system

Parameter Descriptions

Following are descriptions of each of the **parm** file parameters.

[ID](#)

[Log](#)

[Thresholds](#)

[scopetransactions](#)

[subprocinterval](#)

[gapapp](#)

[fstypes](#)

[wait](#)

[Size](#)

[Mainttime](#)

[Days](#)

[Maintweekday](#)

[javaarg](#)

[Flush](#)

[project_app](#)

[proclist](#)

[appproc](#)

[proccmd](#)

[ignore_mt](#)

[cachemem](#)

[kvm_collection_mode](#)

ID

The system ID value is a string of characters that identifies your system. The default ID assigned is the system's hostname. If you want to modify the default ID assigned, make sure all the systems have unique ID strings. This identifier is included in the log files to identify the system on which the data was collected. You can specify a maximum of 39 characters.

Log

The log parameter specifies data types to be collected by scope.

log global enables scope to record global records to the **logglob** file. You must have global data records to view and analyze performance data on your system. Global metrics are not affected by logging options or values of application or process data.

log application enables scope to record active application records to the **logappl** file. By default, scope logs only the applications that have active processes during an interval.

log application=all in the **parm** file enables scope to log all applications to the **logappl** file at every interval, regardless of whether the applications are active or not.

The application=all option may be desirable in specific circumstances in relation to the use of application alarms. For example, you can generate alarm when an application becomes inactive (APP_ALIVE_PROC).

If you enable this option, the log file **logappl** grows in size at a faster rate since all applications are logged at every interval. You can use the **utility** program's scan function to monitor the utilization of the scope log files.

On HP-UX only, you can specify the parameter log application=prm to enable scope to record active Process Resource Manager (PRM) groups to the **logappl** file. If you specify this parameter, scope will not record user-defined application sets listed in the **parm** file. In addition, all application metrics collected will reflect a PRM context and will be grouped by the APP_NAME_PRM_GROUPNAME metric.

Application logging options do not affect global or process data.

log process enables scope to record information about interesting processes to the **logproc** file. A process may become interesting when it is first created, when it ends, and when it exceeds a threshold specified in the parm file for an application. Process threshold logging options have no effect on global or application data.

log device=disk,lvn,cpu,filesystem enables scope to record information about individual disks, logical volumes (HP-UX only), CPUs, and file systems to the **logdev** file.

Note: Do not use lvm if the monitored system does not run with the HP-UX operating system.

By default, only disks, volumes, and interfaces that had I/O generated through them during an interval are logged. netif (logical LAN device) records and disk records (on HP-UX) are always logged regardless of the selected log device options.

For example, to request logging of records for individual disks, logical volumes, CPUs, network interfaces, but *not* individual file systems, use the following setting:

log device=disk , lvm , cpu.

When filesystem is specified, all mounted local file systems are logged at every interval, regardless of the activity.

log device=all in the **parm** file enables scope to log all disk, logical volume, CPU, and network interface devices to the **logdev** file at every interval, regardless of whether the devices are active or not.

If you enable this option, the **logdev** file grows in size at a faster rate since all devices are logged at every interval. Use the **utility** program's scan function to monitor log file utilization and sizing.

log transaction enables scope to record ARM transaction records to the **logtran** file. To enable scope to collect data, a process that is instrumented with the Application Response Measurement (ARM) API must be running on your system. (For more information, see ["What is Transaction Tracking?" on page 336.](#))

The default values for the log transaction parameter are no resource and no correlator.

To enable resource data collection (HP-UX only) or correlator data collection, specify **log transaction=resource** or **log transaction=correlator**. Both can be logged by specifying **log transaction=resource, correlator**.

log logicalsystems enables scope to record information about the logical systems to the **logls** file. Data for logical systems is summarized periodically at intervals specified in the **parm** file.

On AIX 6.1 TL2, BYLS logging for LPAR and WPAR can be configured by using the logicalsystems parameter in the **parm** file. See ["parm File Parameters Used by scope" on page 27.](#)

The log files are created automatically irrespective of logging options. If a particular type of logging is disabled, the corresponding log file will not be removed automatically from the monitored system.

If you specify log without options, scope logs only the global and process data.

Thresholds

The threshold parameters enable scope to record only critical information into the log files and filter out unnecessary, non-critical details of the system.

The following parameters specify the thresholds for different classes of metrics. When the threshold value specified is exceeded for a particular instance of a class of data, a record for that instance is logged by scope.

You can specify lower values for the threshold, to enable scope to log more data or you can specify higher values for the threshold, to enable scope to log lesser data so that you have fewer records logged on average. Listed below are the threshold parameter available:

[Procthreshold](#)

[appthreshold](#)

[diskthreshold](#)

[bynetifthreshold](#)

[fsthreshold](#)

[lvthreshold](#)

[bycputhreshold](#)

Procthreshold

The procthreshold parameter is used to set activity levels to specify criteria for interesting processes. To use this parameter, you must enable process logging. procthreshold affects only processes that are logged and do not affect other classes of data.

You must specify threshold options on the same parameter line (separated by commas).

procthreshold Options for Process Data

cpu	<p>Sets the percentage of CPU utilization that a process must exceed to become “interesting” and be logged.</p> <p>The value percent is a real number indicating overall CPU use. For example, cpu=7.5 indicates that a process is logged if it exceeds 7.5 percent of CPU utilization in a 1-minute sample.</p>
disk	<p>(Not available on Linux or Windows.) Sets the rate of physical disk I/O per second that a process must exceed to become “interesting” and be logged.</p> <p>The value is a real number. For example, disk=8.0 indicates that a process will be logged if the average physical disk I/O rate exceeds 8 KBs per second.</p>
memory	<p>Sets the memory threshold that a process must exceed to become “interesting” and be logged.</p> <p>The value is in megabyte units and is accurate to the nearest 100 KB. If set, the memory threshold is compared with the value of the PROC_MEM_VIRT metric. Each process that exceeds the memory threshold will be logged, similarly to the disk and CPU process logging thresholds.</p>
nonew	<p>Disables logging of new processes if they have not exceeded any threshold. If not specified, all new processes are logged. On HP-UX, if shortlived is <i>not</i> specified, then only new processes that lasted more than one second are logged.</p>
nokilled	<p>Disables logging of exited processes if they did not exceed any threshold. If not specified, all killed (exited) processes are logged. On HP-UX, if shortlived is <i>not</i> specified, then only killed processes greater than one second are logged.</p>
shortlived	<p>Enables logging of processes that ran for less than one second in an interval. (This often significantly increases the number of processes logged.) If scope finds threshold shortlived in the parm file, it logs shortlived processes, regardless of the CPU or disk threshold, as long as the nonew and nokilled options are removed. The default is no shortlived processes will be logged. (Do not specify shortlived in the threshold parameter if you do not want shortlived processes logged.)</p>
process	<p>procthreshold specifies the thresholds for the PROCESS class. The default values are as follows:</p> <p>Processes that used more than 10% of a processor's worth of cpu during the last interval</p>

	Processes had a virtual memory set size over 900 MB
	Processes had an average physical disk I/O rate greater than 5 KB per second

apptreshold

The `apptreshold` parameter is used to specify threshold values for the APPLICATION data class (APP_CPU_TOTAL_UTIL metric). The threshold criteria is based on the percentage of CPU utilization that an application must exceed for the application to be recorded in the log files.

The default setting in the `parm` file enables scope to record applications that use more than 0% of CPU.

diskthreshold

The `diskthreshold` parameter is used to specify the threshold values for DISK class. The threshold criteria for DISK class is based on the percentage of time duration, a disk performs I/Os (BYDSK_UTIL metric).

The default setting in the `parm` file enables scope to record the details of disks that are busy performing I/Os for more than 10% of the time duration.

bynetifthreshold

The `bynetifthreshold` parameter specifies the thresholds for the NETIF class. Netif data class threshold criteria is based on the number of packets transferred by the network interface per second (BYNETIF_PACKET_RATE metric).

The default setting in the `parm` file enables scope to record the details of network interfaces that transfer more than 60 packets per second. If the value for this parameter is not specified or if the parameter is commented out, scope logs the details of all the network interfaces that are not idle.

fsthreshold

The `fsthreshold` parameter specifies the thresholds for FILESYSTEM class. The file system data class threshold criteria is based on the percentage of disk space utilized by the filesystems (FS_SPACE_UTIL metric).

The default setting in the `parm` file enables scope to record the details of filesystems that utilize more than 70% of disk space.

lvthreshold

The `lvthreshold` specifies the thresholds for the LOGICALVOLUME class. Logical volume data class threshold values are based on I/Os per second (LV_READ_RATE + LV_WRITE_RATE).

The default setting in the `parm` file enables scope to record the details of logical volumes that have more than 35 I/Os per second.

bycputhreshold

The `bycputhreshold` parameter specifies the thresholds for CPU class. CPU data class thresholds criteria is based on percentage of time the CPU was busy (`BYCPU_CPU_TOTAL_UTIL`).

The default setting in the **parm** file enables scope to record the details of CPUs that are busy more than 90% of the time.

scopetransactions

The scope collector itself is instrumented with ARM (Application Response Measurement) API calls to log its own transactions. The `scopetransactions` flag determines whether or not scope transactions will be logged. The default is `scopetransactions=on`; scope will log two transactions: `Scope_Get_Process_Metrics` and `Scope_Get_Global_Metrics`. If you do not want these scope transactions to be logged, specify `scopetransactions=off`. A third transaction, `Scope_Log_Headers`, will always be logged; it is not affected by `scopetransactions=off`.

subprocinterval

The `subprocinterval` parameter, if specified, overrides the default that scope uses to sample process data. Process data and global data are logged periodically at intervals specified in the `parm` file. However, scope probes its instrumentation every few seconds to catch short-term activities. This instrumentation sampling interval is 5 seconds by default. The process data logging interval must be an even multiple of the `subprocinterval`. For more information, see ["Configure Data Logging Intervals" on page 52](#).

On some systems with thousands of active threads or processes, the `subprocinterval` should be made longer to reduce overall scope overhead. On other systems with many short-lived processes that you may wish to log, setting the `subprocinterval` lower could be considered, although the effect on scope overhead should be monitored closely in this case. This setting must take values that are factors of the process logging interval as specified in the **parm** file.

Note: The `subprocinterval` parameter is not valid on HP-UX platform.

Lower values for the `subprocinterval` will decrease the gap between global metrics and the sum of applications.

If there is CPU overhead on systems with large number of active processes, it can be avoided by setting the `subprocinterval` value same as the process collection interval value, which is 60.

gapapp

The `gapapp` parameter in the `parm` file controls the modification of application class of data to account for any difference between the global (system-wide) data and the sum of application data.

Application data is derived from process-level instrumentation. Typically there is difference between the global metrics and the sum of applications. In systems which have high process creation rates the difference will be significant. You can choose from the following options:

If `gapapp` is blank, an application named `gapapp` will be added to the application list.

If `gapapp = UnassignedProcesses`, an application by the name `UnassignedProcesses` will be added to the application list.

If `gapapp = ExistingApplicationName(or) gapapp = other`, The difference to the global values will be added to the specified application instead of being logged separately and adding a new entry to the application list.

fstypes

The `fstypes` parameter enables you to monitor specific file systems on the system. By default, the scope collector logs data for all file systems that are attached to the system. With the `fstypes` parameter, you can enable data collection for only specific file systems.

In the **parm** file, you must set the `fstypes` parameter to the name of the file system that you want to monitor. You can specify multiple file system names separated by commas.

This is the syntax of this parameter:

```
fstypes=<file_system1><file_system2>, ...
```

<file_system1> is the file system type.

For HP-UX, Linux, and AIX, available file systems with file system types are listed in the **/etc/fstab** file.

For AIX, available file systems with file system types are listed in the **/etc/filesystems** file.

For Solaris, available file systems with file system types are listed in the **/etc/vfstab** file.

To find out the file system type on Windows, right-click the disk drive in the Windows explorer, and then click **Properties**. The value displayed for File system is the file system type that you can specify with the `fstypes` parameter.

You can set this parameter to an inclusion list or exclusion list. A comma-separated list of file system types indicates an inclusion list and enables the agent to monitor data only for the specified file systems.

```
fstypes = tmpfs, mvfs, nfs
```

Example 1:

While configuring the `fstypes` parameter, use the file system names returned by operating system commands.

You can also set this parameter to an exclusion list, which is a comma-separated list of file system types that begins with the character `!`. Specifying the `!` character in the beginning of the list ensures that the agent does not monitor any file system that belongs to the list.

Example 2:

```
fstypes = !tmpfs, mvfs
```

The above configuration stops monitoring of all temporary file systems and multi-version file systems attached to the monitored node.

If you want to monitor both nfs and nfs3 file systems, you can specify `fstypes = nfs*` instead of `fstypes = nfs, nfs3`

Example 3:

`fstypes =`

or

`fstypes = *`

Specifying `*` or `blank` character ensures that Operations agent will monitor all the file systems that are available.

wait

You can use the `wait` parameter (HP-UX only) to capture details of processes which wait for system resources. You can specify the value of the `wait` parameter in percentage. When a process waits for system resources: `cpu`, `disk`, `mem`, `sem`, and `lan` for a percentage of interval greater than the value specified for the `wait` parameter then the details of that process are logged in the **logproc** file.

See "[parm File Parameters Used by scope](#)" on page 27 for values and options.

For example, if process logging interval is defined as 60 seconds and the `wait` parameter for the CPU is set to 50%, any process waiting for CPU for more than or equal to 30 seconds is captured in the **logproc** file.

Size

The `size` parameter is used to set the maximum size (in megabytes) of any raw log file. You cannot set the size to be less than one megabyte.

The scope collector reads these specifications when it is initiated. If any of these log files achieve their maximum size during collection, they will continue to grow until `mainttime`, when they will be rolled back automatically. During a roll back, the oldest 25 percent of the data is removed from the log file. Raw log files are designed to only hold a maximum of one year's worth of data if not limited by the `size` parameter. See [Log File Contents Summary](#) and [Log File Empty Space Summary](#) in the [Utility Scan Report Details](#).

If the `size` specification in the **parm** file is changed, scope detects it during startup. If the maximum log file size is decreased to the point where existing data does not fit, an automatic resize will take place during the scope startup. If the existing data fits within the new maximum size specified, no action is taken.

The `resize` command creates the new file `scopelog` in the directory set by `TMPDIR` environment variable before deleting the original log file. See [How to Use it](#) section in the [Resize](#).

Any changes you make to the maximum size of a log file take effect at the time specified in the `mainttime` parameter.

Note: Regardless of the `size` parameters, the maximum size of the scope log files will be limited also by the amount of data stored over one year. Raw scope log files cannot contain more than one

year of data, so if logs extend back that long, the data older than one year will be overwritten. See [extract](#) for information about how to create archival log files if more than a year of data is desired.

Mainttime

Log files are rolled back if necessary by scope only at a specific time each day. The default time can be changed using the mainttime parameter. For example, setting **mainttime=8:30**, causes log file maintenance to be done at 8:30 am each day.

We suggest setting mainttime to a time when the system is at its lowest utilization.

Note: Log file maintenance only rolls out data older than one day, so if a log file such as **logproc** grows very quickly and reaches its limit within 24 hours, its size can exceed the configured size limit.

Days

The days parameter specifies the maximum number of days of data, any raw data log file can store at a given point of time. The value for this parameter must be in the range of 1 to 365. This parameter enables scope data collector to maintain log files.

During data collection, if the number of days of data in the log file reaches the days specified in the days parameter, data collection will continue till the day specified in the maintweekday parameter is met. Once maintweekday is reached, the log file will be rolled back automatically at mainttime. During the roll back, data collected after days parameter reached its maximum value will be removed from the log file.

Note: When the log files are rolled back during data collection, if the value specified in the size parameter is reached on a specific day before the days parameter, then the size parameter overrides the days parameter.

Example, if "size global=20" and "days global=40" is used in **parm** file, and if the log files reaches maximum size 20 MB before 40 days of data being logged in log file, then the log file roll back is done based on the size parameter.

Maintweekday

The maintweekday parameter specifies the day of the week when the log file roll back happens if the days parameter is met. The roll back will happen at mainttime.

Example, if "maintweekday=Mon" is used in **parm** file, the log file roll back is done once the value specified in the days parameter is met on "Monday" at mainttime. It is recommended that the value for maintweekday should be set to a day in the week when the system utilization is low.

Note: The maintweekday parameter is an optional parameter. If maintweekday parameter is specified in the **parm** file, it should be used along with the days parameter. This parameter will not be considered, if it is not used with days parameter in the parm file. If maintweekday is not specified in the **parm** file though days parameter is specified, then the default value is "maintweekday=Sun"

Example, if “daysglobal=30”, “application=20”, “process=30”, “device=20”, “transaction=10”, “maintweekday=Wed” and if the log file reaches the number of days specified in the days parameter, data collection will continue till the day specified in the maintweekday. Once maintweekday is reached, log file roll back will happen removing the exceeded number of days of data from the start of the log file. This maintenance will be done at mainttime.

javaarg

Note: This parameter is valid only on UNIX/Linux.

The javaarg parameter is a flag that can be set to true or false. It ONLY affects the value of the proc_proc_argv1 metric.

When javaarg is set to false or is not defined in the **parm** file, the **proc_proc_argv1** metric is always set to the value of the first argument in the command string for the process.

When javaarg is set to true, the proc_proc_argv1 metric is overridden, for Java processes only, with the class or jar specification if that can be found in the command string. In other words, for processes whose file names are java or jre, the proc_proc_argv1 metric is overridden with the first argument without a leading dash not following a -classpath or a -cp, assuming the data can be found in the argument list provided by the OS.

While this sounds complex, it is very plain when you have Java processes running on your system: set **javaarg=true** and the proc_proc_argv1 metric is logged with the class or jar name. This can be very useful if you want to define applications specific to Java. When the class name is in proc_proc_argv1, then you can use the argv1= application qualifier to define your application by class name.

Flush

The flush parameter specifies the data logging intervals (in seconds) at which all instances of application and device data will be logged. The flush intervals must be in the range 300-32700 and be an even multiple of 300.

The default value of flush interval is 3600 seconds for all instances of application and device data.

You can disable the flush parameter by specifying the value as 0 (zero). If the flush parameter is set to 0, scope will not log application and the device data which does not meet the thresholds specified in the **parm** file.

project_app

If you set this parameter to true, the Performance Collection Component deems each Solaris project as an application (and the project ID as the application ID). To ignore Solaris projects, set this parameter to false.

Note: This parameter is supported only for Solaris 10 and above.

proclist

You can use this parameter only in Solaris global zones; it has no effect on the agent that is running in a non-global zone.

In a global zone, if you set this parameter to all, the Performance Collection Component monitors all global and non-global zone processes. To monitor only the processes that belong to the global zone, set this parameter to local.

appproc

This parameter is available only on Solaris. You can use this parameter only in a global zone; it has no effect on the agent that is running in a non-global zone.

In a global zone, if you set this parameter to all, the Performance Collection Component includes the processes for global and non-global zone applications while calculating values of all APP_ metrics. To include only the global zone applications for the calculation of APP_ metrics, set this parameter to local.

proccmd

Note: This parameter is valid only on UNIX/Linux.

The proccmd parameter enables logging of process commands into HP Operations agent data store. You can specify the length of the process command as a numeric value in this parameter. The maximum numeric value is 1024. By default, the value for this parameter is set to 0 and the logging of process commands is disabled.

ignore_mt

If you set this parameter to true, the Performance Collection Component logs all the CPU-related metrics of the Global class after normalizing the metric values against the number of active cores on the monitored system.

When this parameter is set to false, the Performance Collection Component logs all the CPU-related metrics of the Global class after normalizing the metric values against the number of threads on the monitored system.

For Linux machines, this parameter is set to false by default. The Performance Collection Component logs all the CPU-related metrics of the Global class after normalizing the metric values against the number of threads on the monitored system. The Performance Collection Component also logs the metrics of the Core CPU class after normalizing the metric values against the number of active cores on the monitored system.

This parameter has no effect on HP-UX. You must run the **midaemon -ignore_mt** command on HP-UX to switch between the above modes. For more information, see ["Logging Metrics Calculated with the Core-Based Normalization" on page 58](#).

When ignore_mt flag is set as a command-line parameter to midaemon on HP-UX systems, few of the metrics are affected. For more information on the list of metrics which get affected, See ["Metrics affected while setting the ignore_mt flag to midaemon on HP-UX systems" below](#).

The Performance Collection Component ignores this parameter if the multithreading property is disabled on the system. As a result, the value of the GBL_IGNORE_MT metric is logged as true.

Note: If you enable or disable Simultaneous Multi-Threading (SMT) on a Windows, Linux, or Solaris system, you must restart the system.

Metrics affected while setting the ignore_mt flag to midaemon on HP-UX systems

Global Metric Class	
GBL_CPU_TOTAL_UTIL	GBL_CPU_CSWITCH_UTIL_CUM
GBL_CPU_TOTAL_UTIL_CUM	GBL_CPU_CSWITCH_UTIL_HIGH
GBL_CPU_TOTAL_UTIL_HIGH	GBL_CPU_CSWITCH_TIME
GBL_CPU_TOTAL_TIME	GBL_CPU_CSWITCH_TIME_CUM
GBL_CPU_TOTAL_TIME_CUM	GBL_CPU_INTERRUPT_UTIL
GBL_CPU_SYS_MODE_UTIL	GBL_CPU_INTERRUPT_UTIL_CUM
GBL_CPU_SYS_MODE_UTIL_CUM	GBL_CPU_INTERRUPT_UTIL_HIGH
GBL_CPU_SYS_MODE_TIME	GBL_CPU_INTERRUPT_TIME
GBL_CPU_SYS_MODE_TIME_CUM	GBL_CPU_INTERRUPT_TIME_CUM
GBL_CPU_TRAP_TIME	GBL_CPU_VFAULT_TIME
GBL_CPU_TRAP_TIME_CUM	GBL_CPU_VFAULT_TIME_CUM
GBL_CPU_TRAP_UTIL	GBL_CPU_VFAULT_UTIL
GBL_CPU_TRAP_UTIL_CUM	GBL_CPU_VFAULT_UTIL_CUM
GBL_CPU_TRAP_UTIL_HIGH	GBL_CPU_VFAULT_UTIL_HIGH
GBL_CPU_USER_MODE_TIME	GBL_CPU_IDLE_UTIL
GBL_CPU_USER_MODE_TIME_CUM	GBL_CPU_IDLE_UTIL_CUM
GBL_CPU_USER_MODE_UTIL	GBL_CPU_IDLE_UTIL_HIGH
GBL_CPU_USER_MODE_UTIL_CUM	GBL_CPU_IDLE_TIME
GBL_CPU_NICE_UTIL	GBL_CPU_IDLE_TIME_CUM
GBL_CPU_NICE_UTIL_CUM	GBL_CPU_NORMAL_UTIL
GBL_CPU_NICE_UTIL_HIGH	GBL_CPU_NORMAL_UTIL_CUM
GBL_CPU_NICE_TIME	GBL_CPU_NORMAL_UTIL_HIGH

GBL_CPU_NICE_TIME_CUM	GBL_CPU_NORMAL_TIME
GBL_CPU_NNICE_UTIL	GBL_CPU_NORMAL_TIME_CUM
GBL_CPU_NNICE_UTIL_CUM	GBL_CPU_SYSCALL_UTIL
GBL_CPU_NNICE_UTIL_HIGH	GBL_CPU_SYSCALL_UTIL_CUM
GBL_CPU_NNICE_TIME	GBL_CPU_SYSCALL_UTIL_HIGH
GBL_CPU_NNICE_TIME_CUM	GBL_CPU_SYSCALL_TIME
GBL_CPU_REALTIME_UTIL	GBL_CPU_SYSCALL_TIME_CUM
GBL_CPU_REALTIME_UTIL_CUM	GBL_CPU_WAIT_UTIL
GBL_CPU_REALTIME_UTIL_HIGH	GBL_CPU_WAIT_TIME
GBL_CPU_REALTIME_TIME	GBL_CPU_WAIT_TIME_CUM
GBL_CPU_REALTIME_TIME_CUM	GBL_CPU_WAIT_UTIL_CUM
GBL_CPU_CSWITCH_UTIL	GBL_CPU_WAIT_UTIL_HIGH
Application Metric Class	
APP_PRM_CPU_TOTAL_UTIL_CUM	APP_CPU_NORMAL_UTIL
APP_CPU_NNICE_UTIL	APP_CPU_USER_MODE_UTIL
APP_CPU_NNICE_TIME	APP_CPU_TOTAL_TIME
APP_CPU_TOTAL_UTIL	APP_CPU_SYS_MODE_TIME
APP_CPU_TOTAL_UTIL_CUM	APP_CPU_NICE_TIME
APP_CPU_SYS_MODE_UTIL	APP_CPU_REALTIME_TIME
APP_CPU_NICE_UTIL	APP_CPU_NORMAL_TIME
APP_CPU_REALTIME_UTIL	APP_CPU_USER_MODE_TIME
PROC Metric Class	
PROC_CPU_TOTAL_UTIL	PROC_CPU_REALTIME_UTIL
PROC_CPU_TOTAL_UTIL_CUM	PROC_CPU_REALTIME_UTIL_CUM
PROC_CPU_TOTAL_TIME	PROC_CPU_REALTIME_TIME
PROC_CPU_TOTAL_TIME_CUM	PROC_CPU_REALTIME_TIME_CUM
PROC_CPU_SYS_MODE_UTIL	PROC_CPU_CSWITCH_UTIL
PROC_CPU_SYS_MODE_UTIL_CUM	PROC_CPU_CSWITCH_UTIL_CUM

PROC_CPU_SYS_MODE_TIME	PROC_CPU_CSWITCH_TIME
PROC_CPU_SYS_MODE_TIME_CUM	PROC_CPU_CSWITCH_TIME_CUM
PROC_CPU_USER_MODE_UTIL	PROC_CPU_INTERRUPT_UTIL
PROC_CPU_USER_MODE_UTIL_CUM	PROC_CPU_INTERRUPT_UTIL_CUM
PROC_CPU_USER_MODE_TIME	PROC_CPU_INTERRUPT_TIME
PROC_CPU_USER_MODE_TIME_CUM	PROC_CPU_INTERRUPT_TIME_CUM
PROC_CPU_NICE_UTIL	PROC_CPU_NORMAL_UTIL
PROC_CPU_NICE_UTIL_CUM	PROC_CPU_NORMAL_UTIL_CUM
PROC_CPU_NICE_TIME	PROC_CPU_NORMAL_TIME
PROC_CPU_NICE_TIME_CUM	PROC_CPU_NORMAL_TIME_CUM
PROC_CPU_NNICE_UTIL	PROC_CPU_SYSCALL_UTIL
PROC_CPU_NNICE_UTIL_CUM	PROC_CPU_SYSCALL_UTIL_CUM
PROC_CPU_NNICE_TIME	PROC_CPU_SYSCALL_TIME
PROC_CPU_NNICE_TIME_CUM	PROC_CPU_SYSCALL_TIME_CUM
PROC_CPU_ALIVE_TOTAL_UTIL	
PROC_CPU_ALIVE_USER_MODE_UTIL	
PROC_CPU_ALIVE_SYS_MODE_UTIL	
BYCPU Metric Class	
BYCPU_CPU_TOTAL_UTIL	BYCPU_CPU_INTERRUPT_TIME
BYCPU_CPU_TOTAL_UTIL_CUM	BYCPU_CPU_INTERRUPT_TIME_CUM
BYCPU_CPU_TRAP_TIME	BYCPU_CPU_INTERRUPT_UTIL
BYCPU_CPU_TRAP_TIME_CUM	BYCPU_CPU_INTERRUPT_UTIL_CUM
BYCPU_CPU_TRAP_UTIL	BYCPU_CPU_CSWITCH_TIME
BYCPU_CPU_TRAP_UTIL_CUM	BYCPU_CPU_CSWITCH_TIME_CUM
BYCPU_CPU_USER_MODE_TIME	BYCPU_CPU_CSWITCH_UTIL
BYCPU_CPU_USER_MODE_TIME_CUM	BYCPU_CPU_CSWITCH_UTIL_CUM
BYCPU_CPU_USER_MODE_UTIL	BYCPU_CPU_VFAULT_TIME
BYCPU_CPU_USER_MODE_UTIL_CUM	BYCPU_CPU_VFAULT_TIME_CUM

BYCPU_CPU_NICE_TIME	BYCPU_CPU_VFAULT_UTIL
BYCPU_CPU_NICE_TIME_CUM	BYCPU_CPU_VFAULT_UTIL_CUM
BYCPU_CPU_NICE_UTIL	BYCPU_CPU_REALTIME_TIME
BYCPU_CPU_NICE_UTIL_CUM	BYCPU_CPU_REALTIME_TIME_CUM
BYCPU_CPU_NNICE_TIME	BYCPU_CPU_REALTIME_UTIL
BYCPU_CPU_NNICE_TIME_CUM	BYCPU_CPU_REALTIME_UTIL_CUM
BYCPU_CPU_NNICE_UTIL	BYCPU_CPU_NORMAL_TIME
BYCPU_CPU_NNICE_UTIL_CUM	BYCPU_CPU_NORMAL_TIME_CUM
BYCPU_CPU_TOTAL_TIME	BYCPU_CPU_NORMAL_UTIL
BYCPU_CPU_TOTAL_TIME_CUM	BYCPU_CPU_NORMAL_UTIL_CUM
BYCPU_CPU_SYS_MODE_TIME	BYCPU_CPU_SYSCALL_TIME
BYCPU_CPU_SYS_MODE_TIME_CUM	BYCPU_CPU_SYSCALL_TIME_CUM
BYCPU_CPU_SYS_MODE_UTIL	BYCPU_CPU_SYSCALL_UTIL
BYCPU_CPU_SYS_MODE_UTIL_CUM	BYCPU_CPU_SYSCALL_UTIL_CUM
SYSCALL Metric Class	
SYSCALL_CPU_TOTAL_TIME	SYSCALL_CPU_TOTAL_TIME_CUM

cachemem

The `cachemem` parameter in the `parm` file enables you to configure the agent to include the buffer cache size while reporting the total memory utilization data.

You can set the `cachemem` parameter to *one* of the following:

1. **Set the `cachemem` parameter to free (f).**

By default, the parameter is set to free (f). The following metric values are affected:

GBL_MEM_UTIL- Does not include the buffer cache size.

GBL_MEM_FREE_UTIL - Includes the buffer cache size.

2. **Set the `cachemem` parameter to user (u).**

The following metric values are affected:

GBL_MEM_UTIL- Includes the buffer cache size.

GBL_MEM_FREE_UTIL - Does not include the buffer cache size.

Note:

On AIX machines, you should set the `cachemem` parameter to **free (f)** to match with `topas` commands.

On Solaris machines, `cachemem` parameter is applicable only for Adaptive Replacement Cache (ARC) for ZFS. `Cachemem` parameter is set to **user (u)** by default. You can set this parameter to **free (f)** in the `parm` file and then restart the data collector. `GBL_MEM_UTIL` excludes ZFS ARC cache when `cachemem` parameter in the `parm` file is set to **free**.

kvm_collection_mode

The `kvm_collection_mode` parameter enables the root and non-root user to view either the system or session virtual machines depending on the value set for the parameter. The default value for this parameter is `session`. You can set this parameter value either to `session` or `system` in the `parm` file.

Note: The `kvm_collection_mode` parameter is valid only on Linux systems.

The `kvm_collection_mode` parameter can be set in the following modes:

- **Non-root mode:** If the `kvm_collection` parameter is set to `system`, the non-root user can access only system virtual machines and if the `kvm_collection_mode` parameter is set to `session`, the non-root user can access only the session virtual machines.

The SUID permission for Glance allows non-root users to access Glance with root privileges. If SUID permission is set for Glance the non-root users cannot access the session virtual machines and may see the following error message:

```
No data Available
```

- **Root mode:** The root user can access only system virtual machines irrespective of the values set for the `kvm_collection_mode` parameter.

Application Definition Parameters

The following parameters pertain to application definitions: `application`, `file`, `user`, `group`, `cmd`, `argv1`, or `priority`.

The Performance Collection Component groups logically related processes together into an application to log the combined effect of the processes on computing resources such as memory and CPU.

Note: In PRM mode (for HP-UX only), active PRM groups are logged and the user-defined application sets listed in the `parm` file are ignored.

An application can be a list of files (base program names), a list of commands, or a combination of these also qualified by user names, group names, or argument selections. All these application qualifiers can be used individually or in conjunction with each other. If, for example, `cmd` and `user` qualifiers are both used then a process must meet the specification of both the command string and the user name in order to belong to that application. Each qualifier is discussed in detail below.

Note: Any process on the system belongs to only one application. No process is counted into two or more applications.

Application

The application name defines an application or class that groups together multiple processes and reports on their combined activities.

The application name is a string of up to 19 characters used to identify the application.

Application names can be lowercase or uppercase and can contain letters, numbers, underscores, and embedded blanks. Do not use the same application name more than once in the **parm** file.

An equal sign (=) is optional between the application keyword and the application name.

The application parameter must precede any combination of `file`, `user`, `group`, `cmd`, `argv1`, and or parameters that refer to it, with all such parameters applying against the last application workload definition.

Each parameter can be up to 170 characters long including the carriage return character, with no continuation characters permitted. If your list of files is longer than 170 characters, continue the list on the next line after another `file`, `user`, `group`, `cmd` or `argv1` statement.

You can define up to 998 applications. Performance Collection Component predefines an application named `other`. The `other` application collects all processes not captured by `application` statements in the parm file.

For example:

```
application Prog_Dev
file vi,cc,ccom,pc,pascomp,dbx,xdb
```

```
application xyz
file xyz*,startxyz
```

You can have a maximum of 4096 `file`, `user`, `group`, `argv1`, and `cmd` specifications for all applications combined. The previous example includes nine file specifications. (`xyz*` counts as only one specification even though it can match more than one program file.)

If a program file is included in more than one application, it is logged in the first application that contains it.

The default **parm** file contains some sample applications that you can modify. The **examples** directory also contains other samples (in a file called **parm_apps**) you can copy into your parm file and modify as needed.

File

The file parameter specifies the program files that belong to an application. All interactive or background executions of these programs are included. It applies to the last application statement issued. An error is generated if no application statement is found.

The file name can be any of the following:

- For UNIX/Linux:
 - A single UNIX program file such as `vi`.
 - A group of UNIX program files (indicated with a wild card) such as `xyz*`. In this case, any program name that starts with the letters `xyz` is included. A file specification with wild cards counts as only one specification toward the maximum allowed.
- For Windows:
 - A single program file such as `winword`.
 - A group of program files (indicated with a wild card) such as `xyz*`. In this case, any program name that starts with the letters `xyz` is included. A file specification with wild cards counts as only one specification toward the maximum of 1000 for all file specifications.

Note: For Windows, when you define executable files for an application in the `parm` file, no file extensions are required. For example, you can define `winword` in the `parm` file without its `.exe` extension.

The name in the file parameter is limited to 15 characters in length. An equal sign (=) is optional between the file parameter and the file name.

You can enter multiple file names on the same parameter line (separated by commas) or in separate file statements. File names cannot be qualified by a path name. The file specifications are compared to the specific metric `PROC_PROC_NAME`, which is set to a process's `argv[0]` value (typically its base name). For example:

For UNIX/Linux:

```
application = prog_dev
file = vi,vim,gvim,make,gmake,lint*,cc*,gcc,ccom*,cfront
file = cpp*,CC,cpass*,c++*
file = xdb*,adb,pxdb*,dbx,xlC,ld,as,gprof,lex,yacc,are,nm,gencat
file = javac,java,jre,aCC,ctcom*,awk,gawk
```

```
application Mail
file = sendmail,mail*,*mail,elm,xmh
```

For Windows:

```
application payroll
file account1,basepay,endreport
application Office
file winword* excel*
file 123* msaccess*
```

If you do not specify a file parameter, all programs that satisfy the other parameters qualify.

Note: The asterisk (*) is the only wild card character supported for `parm` file application qualifiers except for the `cmd` qualifier (see below).

argv1

The argv1 parameter specifies the processes that are selected for the application by the value of the PROC_PROC_ARGV1 metric. This is normally the first argument of the command line, except when javaarg=true, when this is the class or jar name for Java processes. This parameter uses the same pattern matching syntax used by **parm** parameters like **file=** and **user=**. Each selection criteria can have asterisks as a wildcard match character, and you can have more than one selection on one line separated by commas.

For example, the following application definition buckets all processes whose first argument in the command line is either -title, -fn, or -display:

```
application = xapps  
argv1 = -title,-fn,-display
```

The following application definition buckets a specific Java application (when javaarg=true):

```
application = JavaCollector  
argv1 = com.*Collector
```

The following shows how the argv1 parameter can be combined with the file parameter:

```
application = sun-java  
file = java  
argv1 = com.sun*
```

cmd

The cmd parameter specifies processes for inclusion in an application by their command strings, which consists of the program executed and its arguments (parameters). Unlike other selection parameters, this parameter allows extensive use of wildcard characters besides the use of the asterisk character.

Similar to regular expressions, extensive pattern matching is allowed. For a complete description of the pattern criteria, see the UNIX man page for fnmatch. Unlike other parameters, you can have only one selection per line, however you can have multiple lines.

The following shows use of the cmd parameter:

```
application = newbie  
cmd = *java *[Hh]ello[Ww]orld*
```

Note: The cmd parameter is supported only on Unix/Linux systems.

User

The user parameter specifies which users (login names) belong to the application. The format is as below:

```
application <application_name>  
file <file_name>  
user [<Domain_Name>]\<User_Name>
```

The domain name in the user parameter is optional. You must specify the domain name to specify the user names of a non-local system.

For example:

```
application test_app
file test
user TestDomain\TestUser
```

If you specify the user name without the domain name in the user parameter, the user names will be considered to be the user names of the local system.

For example:

```
application Prog_Dev
file vi,xb,abb,ld,lint
user ted,rebecca,test*
```

You can only use the wild card asterisk (*) to ensure the user names with a similar string of characters prefixed before the asterisk (*) and suffixed after the asterisk (*) belong to the application. If you do not specify a user parameter, all programs that satisfy the other parameters qualify.

The name in the user parameter is limited to 15 characters in length.

Group

The group parameter specifies which user group names belong to an application.

For example:

```
application Prog_Dev_Group2
file vi,xb,abb,ld,lint
user ted,rebecca,test*
group lab, test
```

If you do not specify a group parameter, all programs that satisfy the other parameters qualify.

The name in the group parameter is limited to 15 characters in length.

Or

Use the or parameter to allow more than one application definition to apply to the same application. Within a single application definition, a process must match at least one of each category of parameters. Parameters separated by the or parameter are treated as independent definitions. If a process matches the conditions for any definition, it will belong to the application.

For example:

```
application = Prog_Dev_Group2
user julie
or
user mark
file vi, store, dmp
```

This defines the application (Prog_Dev_Group2) that consists of any programs run by the user julie plus other programs (vi, store, dmp) if they are executed by the user mark.

Priority

You can restrict processes in an application to those belonging to a specified range by specifying values in the priority parameter.

For example:

```
application = swapping  
priority 128-131
```

Processes can range in priority from -511 to 255, depending on which platform the HP Operations agent is running. The priority can be changed over the life of a process. The scheduler adjusts the priority of time-share processes. You can also change priorities programmatically or while executing.

Note: The **parm** file is processed in the order entered and the first match of the qualifier will define the application to which a particular process belongs. Therefore, it is normal to have more specific application definitions prior to more general definitions.

Application Definition Examples

The following examples show application definitions.

```
application firstthreesvrs  
cmd = *appserver* *-option[123]*
```

```
application oursvrs  
cmd = *appserver*  
user = xyz,abc
```

```
application othersvrs  
cmd = *appserver*  
cmd = *appsvr*  
or  
argv1 = -xyz
```

The following is an example of how several of the programs would be logged using the preceding **parm** file.

Command String	User Login	Application
/opt/local/bin/appserver -xyz -option1	xyz	firstthreesvrs
./appserver -option5	root	othersvrs
./appserver -xyz -option2 -abc	root	firstthreesvrs
./appsvr -xyz -option2 -abc	xyz	othersvrs

Command String	User Login	Application
<code>./appclient -abc</code>	root	other
<code>./appserver -mno -option4</code>	xyz	oursvrs
<code>appserver -option3 -jkl</code>	xyz	firstthreesvrs
<code>/tmp/bleh -xyz -option1</code>	xyz	othersvrs

Configure Data Logging Intervals

The default collection intervals used by scope are 60 seconds for process data and 300 seconds for global and all other classes of data. You can override this using the `collectioninterval` parameter in the **parm** file. The values must satisfy the following conditions:

The collection intervals for process data can be configured between 5 to 60 seconds in steps of 5 seconds. The collection intervals for process data must be a multiple of the `subprocinterval` (see [subprocinterval](#)) and it must divide evenly into the global collection interval.

The collection interval for global data can be configured to one of the following values: 15, 30, 60 and 300 seconds. The global collection interval must be greater than or equal to process interval, and a multiple of the process collection interval. The global collection interval applies to the global metrics and all non-process metric classes such as filesystem and application.

Configuring Data Collection on vMA Nodes

The HP Operations agent uses the `viserver` daemon to log data on the vMA system. You can configure `viserver` settings in the following configuration files available at `/var/opt/perf`:

`viserver.properties`

`VILog4j.xml`

Note: This version of HP Operations agent does not support IPv6 in vMA.

Configuring Data Collection for Frames

You can collect performance data from all AIX LPARs available on a single frame by installing the HP Operations Agent on only one LPAR node. You can also configure the agent to collect performance data from the AIX frame where all the monitored LPARs reside.

You can have the following advantages if you enable the monitoring of frames:

- Collect configuration information:
 - Name and UUID of the frame where monitored LPARs reside
 - Model, serial number, and type of the frame

- CPU configuration and memory capacity of the frame
- With the additional information, you can analyze resource utilization of the frame
- Use data analysis tools (like HP Performance Manager) to analyze the CPU consumption ratio of the frame and all the LPARs

Task 1: Configure Passwordless SSH Access

On the Hardware Management Console (HMC) system, create a new user, and then, for the newly created user, configure passwordless SSH access between the LPAR node where you installed the agent and the HMC system. As a result, the user must be able to run commands using the SSH protocol on the HMC system remotely from the LPAR node without having to provide a password for authentication.

Verify that, with the newly created user, you can run commands on the HMC system from the LPAR node.

Follow these steps to run a sample command:

1. Log on to the LPAR node as root.
2. Run the following command:

```
ssh hmcuser@hmchost.example.domain.com lssyscfg r sys
```

If the command shows the list of frames without prompting for password, the user is correctly configured and is ready to be used for frame utilization monitoring.

Task 2: Enable Frame Utilization Monitoring on the HMC System

1. Log on to the HMC system as root.
2. Run the following command:

```
chlparutil -r config -s 60
```

Task 3: Configure the HP Operations Agent

1. Log on to the LPAR node.
2. Go to the `/var/opt/perf` directory.
3. Create a new file.
4. Open the file with a text editor, and then add the following content to the file:

```
<hmcusername>@<hmc_fqdn>
```

In this instance:

<hmcusername> is the user that was granted passwordless SSH access to the HMC system in ["Task 1: Configure Passwordless SSH Access"](#) above.

<hmc_fqdn> is the fully qualified domain name of the HMC host.

For example:

```
hmcusername@hmchost.example.domain.com
```

5. Save the file as `hmc` in the `/var/opt/perf` directory.
6. Configure the `logicalsystem` parameter in the `parm` file and start the collection process. For more information about configuring the `parm` file, see ["parm File" on page 24](#).
7. Restart the Performance Collection Component.

The HP Operations Agent collects the frame-specific performance data at the interval of five minutes.

Enabling the Global and Process System Call Metrics for GlancePlus on Linux

On a Linux machine, you can enable the collection of `GBL_SYSCALL`, `SYSCALL`, and `PROCSYSCALL` performance metric values for GlancePlus. Before you start configuring the metric collection, make sure to stop all the processes that use `FTRACE` on your Linux machine.

Note: The metrics are supported for RHEL/OEL/CentOS 6.1 and above, SLES 11 SP1 and above, Ubuntu 11.10 and above, Debian 6.0 and above.

You can enable the metric collection for GlancePlus by *one* of the following methods:

- [Configuring the Metric Collection using `init_ftrace.sh`](#)
- [Configuring the Metric Collection using manual steps](#)

Configuring the Metric Collection using `init_ftrace.sh`

To configure the metric collection on your Linux machine using `init_ftrace.sh`, follow these steps:

1. Log on to the node with the necessary privileges.
2. To stop all the Performance Collection Component processes on your Linux machine, run the following command:

```
/opt/perf/bin/ovpa stop all
```

3. Stop all `midaemon` processes on your Linux machine. To check the `midaemon` process, run the following command:

```
ps -ef | grep midaemon
```

If `midaemon` process is running on your Linux machine, run the following command:

```
killall midaemon
```

This stops all `midaemon` processes.

4. To configure the metric collection, run the following command:

```
/opt/perf/bin/init_ftrace.sh
```

The command mounts `debugfs` and enables FTRACE on your Linux machine. If FTRACE is already running on your machine with any other application, message appears as:

Do you want to reset the FTRACE interface for use with midaemon? (Y/N).

You can select *one* of the following:

Y – Resets the FTRACE interface on your Linux machine.

N – Does not configure the metric collection. You cannot view the metrics in GlancePlus.

5. To start all the Performance Collection Component processes along with the `midaemon` process and metric collection, run the following command:

`/opt/perf/bin/ovpa start all`

Use the following screens in GlancePlus to check the performance metrics:

Y - to check the SYSCALL metrics

L - to check the PROCSYSCALL metrics

Note: When GlancePlus is not at System Call screen, the performance of the system is not affected even if `debugfs` is mounted and FTRACE is enabled. To unmount `debugfs` from `/sys/kernel/debug`, you can run the command **`umount /sys/kernel/debug`**.

Configuring the Metric Collection using manual steps

To configure the metric collection manually on your Linux machine, follow these steps:

1. Log on to the node with the necessary privileges.
2. To stop all the Performance Collection Component processes on your Linux machine, run the following command:

`/opt/perf/bin/ovpa stop all`

3. Stop all `midaemon` processes on your Linux machine. To check the `midaemon` process, run the following command:

`ps -ef | grep midaemon`

If `midaemon` process is running on your Linux machine, run the following command:

`killall midaemon`

This stops all `midaemon` processes.

4. **Mount `debugfs` manually on your Linux machine**

Note: Skip this step if `debugfs` is already mounted on your Linux machine. To check if `debugfs` is already mounted, run the following command:

`cat /proc/mounts |grep debugfs`

The command generates *one* of the following outputs:

`nodev /sys/kernel/debug debugfs rw,relatime 0 0` - `debugfs` is already mounted on your machine.

`nil` – `debugfs` is not mounted on your machine.

To mount debugfs manually, follow these steps:

- i. Run the following command:

```
mount -t debugfs nodev /sys/kernel/debug
```

The output appears as

nil - debugfs is successfully mounted on your machine.

If debugfs is not successfully mounted on your machine, a failure message appears.

5. **Enable FTRACE manually on your Linux machine**

Note: Skip this step if FTRACE is already enabled on your Linux machine. To check if FTRACE is already enabled, run the following command:

```
cat /proc/sys/kernel/ftrace_enabled
```

The command generates *one* of the following outputs:

1 - FTRACE is already enabled on your machine.

0 - FTRACE is not enabled on your machine.

To enable FTRACE manually, follow the step:

- i. Run the following command:

```
echo "1" >/proc/sys/kernel/ftrace_enabled
```

The output appears as

nil - FTRACE is successfully enabled on your machine

If FTRACE is not successfully enabled on your machine, a failure message appears.

6. To start all the Performance Collection Component processes along with the `midaemon` process and metric collection, run the following command:

```
/opt/perf/bin/ovpa start all
```

Use the following screens in GlancePlus to check the performance metrics:

Y - to check SYSCALL metrics

L - to check the PROCSYSCALL metrics

Note: When GlancePlus is not at System Call screen, the performance of the system is not affected even if debugfs is mounted and FTRACE is enabled. To unmount debugfs from `/sys/kernel/debug`, you can run the command **umount /sys/kernel/debug**.

Troubleshooting

This section describes the solution or workaround for the common problem encountered while configuring the metric collection.

Problem:

If a `midaemon` process error occurs, see the `/var/opt/perf/status.mi` for more information. You may experience `midaemon` process error if the `midaemon` has run out of room in its shared memory segment.

Solution:

To avoid this problem, you have to clear the unwanted shared memory. To clear the shared memory, follow these steps:

1. Run the following command:

```
ipcs -m | grep 0x0c6629c9
```

The command will generate an output with a variable value in the second data field. To clear the shared memory, run the following command:

```
ipcrm -m <field_value>
```

Example

The following example shows how to clear the shared memory:

```
ipcs -m | grep 0x0c6629c9  
output= 0x0c6629c9 18841617 root 640 8704448 7  
ipcrm -m 18841617
```

2. To restart mid daemon, run the following command:

```
/opt/perf/bin/ovpa restart
```

Collection of BYCORE and THREAD SYSCALL metric values in advisory mode on Linux

On a Linux machine, you can collect the BYCORE and THREAD SYSCALL performance metric values in advisory mode for GlancePlus.

Note: The metrics are supported for RHEL/OEL/CentOS 6.1 and above, SLES 11 SP1 and above, Ubuntu 11.10 and above, Debian 6.0 and above.

Collection of BYCORE metric values in advisory mode

Run the following command to collect the BYCORE performance metric values in advisory mode for GlancePlus:

```
/opt/perf/bin/glance -aos/tmp/bycore.adv
```

Note: Make sure the adviser script is available under **tmp** folder and is named as **bycore.adv**.

Collection of THREAD SYSCALL metric values in advisory mode

Run the following command to collect the THREAD SYSCALL performance metric values in advisory mode for GlancePlus:

```
/opt/perf/bin/glance -aos/tmp/threadsyscall.adv
```

Note: Before you start the metric collection, make sure FTRACE is enabled in the Linux machine. The adviser script is available under **tmp** folder and is named as **threadsyscall.adv**.

Normalizing CPU Metrics on Hyper-Threading/Simultaneous Multi-Threading-Enabled Systems

On a system where hyper-threading/simultaneous multi-threading (HT/SMT) is enabled, the physical CPU supports two or more hardware threads. As a result, multiple software processes or threads can run on the hardware threads simultaneously. On a system with a multi-core processor, multiple threads can run simultaneously on individual cores.

The Performance Collection Component provides you with several CPU-related metrics, which help you analyze and understand the CPU utilization of the monitored system. By default, on all HT/SMT-enabled systems, the Performance Collection Component calculates the values of all CPU-related metrics by normalizing the gathered data against the number of available threads on the monitored system. When a single thread completely utilizes the entire CPU core, values calculated using the **thread-based normalization** do not always represent the true picture of the CPU utilization.

This version of the HP Operations agent introduces a new configuration parameter, `ignore_mt`, which enables you to configure the Performance Collection Component to log the CPU-related data that has been calculated using the **core-based normalization**. Metric values that are calculated with the core-based normalization present a more accurate status of the CPU utilization, and therefore, help you make more effective decisions while analyzing the system's performance.

Logging Metrics Calculated with the Core-Based Normalization

On HP-UX, you can configure the Performance Collection Component to log all CPU-related metrics with core-based normalization. On other platforms, you can configure the Performance Collection Component to calculate the CPU-related metrics of the GLOBAL class using the core-based normalization before logging.

To configure the Performance Collection Component to use the core-based normalization for CPU-related metrics, follow these steps:

On HP-UX

1. Log on to the system with the root privileges.
2. Configure the `parm` file based on your requirement. Do not set the `ignore_mt` flag in the `parm` file.

Note: The value of the `ignore_mt` flag in the `parm` file on HP-UX has no effect on the operation of the Performance Collection Component.

3. Define alarm rules as necessary.

4. Run the following command:

```
/opt/perf/bin/midaemon -ignore_mt
```

5. Start the HP Operations agent by running the following command:

```
/opt/OV/bin/opcagt -start
```

The Performance Collection Component starts logging all CPU-related metrics (for all classes) using the core-based normalization.

If you restart the HP Operations agent, the Performance Collection Component starts logging the CPU data with the thread-based normalization again and you must configure the Performance Collection Component once again by using the above steps. To enable the agent to always use the core-based normalization, follow these steps:

1. On the agent node, go to the following location:

```
/var/opt/perf
```

2. Open the following file with a text editor:

```
vppa.env
```

3. Set the MIPARMS parameter to ignore_mt.
4. Save the file.
5. Restart the agent by running the following command:

```
/opt/OV/bin/opcagt -start
```

On other platforms

1. Log on to the system with the root or administrative privileges.
2. Configure the parm file based on your requirement. Set the ignore_mt flag in the parm file to true.
3. Define alarm rules as necessary.
4. Start the HP Operations agent using the following command:

On Windows

```
%ovinstalldir%bin\opcagt -start
```

On HP-UX, Linux, or Solaris

```
/opt/OV/bin/opcagt -start
```

On AIX

```
/usr/lpp/OV/bin/opcagt -start
```

The Performance Collection Component starts logging CPU-related metrics for the GLOBAL class using the core-based normalization.

Stopping and Restarting Data Collection

The scope collector and the other associated processes are designed to run continuously. The only time you should stop them are when any of the following occurs:

You are updating Performance Collection Component software to a new release.

You are adding or deleting transactions in the transaction configuration file, **ttd.conf**. (For more information, see [What is Transaction Tracking?](#))

You are modifying distribution ranges or service level objectives (SLOs) in the transaction configuration file, **ttd.conf**. (For more information, see [What is Transaction Tracking?](#))

You are changing the **parm** file and want the changes to take effect. Changes made to the **parm** file take effect only when scope is started.

You are using the **utility** program's **resize** command to resize a Performance Collection Component log file.

You are shutting down the system.

You are adding the hardware or modifying the configuration changes. Changes made will take effect only when scope is started.

Stopping Data Collection

The **ovpa** and **mwa** script's **stop** option ensures that no data is lost when scope and other Performance Collection Component processes are stopped.

To manually stop data collection, type the following command:

On Windows:

```
%ovinstalldir%\bin\ovpacmd stop
```

On HP-UX, Linux, and Solaris:

```
/opt/perf/bin/ovpa -stop
```

On AIX:

```
/usr/lpp/perf/bin/ovpa -stop
```

Note: **scope** does not log NFS data but you can view the NFS data through GlancePlus on the local file system.

Restarting Data Collection

You have different options for restarting data collection after the Performance Collection Component processes have stopped or configuration files have been changed and you want these changes to take effect.

To start scope and the other Performance Collection Component processes after the system has been down, or after you have stopped them, use **<InstallDir>/ovpa start**. Here, **InstallDir** is the directory where Performance Collection Component is installed.

To restart scope and the other processes while they are running, use **<InstallDir>/ovpa restart**. Here, **InstallDir** is the directory where Performance Collection Component is installed. This stops the currently running processes and starts them again.

When you restart scope, the Performance Collection Component continues to use the same log files (**logglob**, **logappl**, **logproc**, **logdev**, **logtran**, **logls**, and **logindx**) used before stopping the program. New records are appended to the end of the existing files. If you want to collect data to a new set of files, and not retain any historical information, you should rename or archive, and remove *all* the scope log files together before you restart, because data is synchronized among the files.

Note: The SEM_KEY_PATH entry in the **ttd.conf** configuration file is used for generating IPC keys for the semaphores used in ttd and the midaemon process on UNIX platforms. The default value used is /var/opt/perf/datafiles. You can change the value of SEM_KEY_PATH if midaemon or ttd does not respond because of sem id collisions.

Daylight Savings

During daylight savings, the system time is set back by one hour in relevant time zones. At this point, data collection stops for an hour until the system time synchronizes with the timestamp of the last logged record.

When daylight savings is turned off, the system time advances by one hour, and therefore, the timestamp of the next logged record advances by an hour. This introduces a one-hour gap after the last logged record even though data collection does not stop.

Changing System Time Manually

When the system time is set back manually, data collection stops and commands (like **perfstat**) do not work. These utilities hang when the system time is set back. To continue logging data and to get responses from all commands, perform the following steps:

Run the following command:

```
ovc -stop coda
```

Back up the **coda.*** files in the `<DataDir>\datafiles\` directory and remove them.

Run the following command:

```
ovc -start coda
```

Note: The system time must synchronize with the data collection time to avoid any gaps in the collection.

Effective Data Collection Management

Efficient analysis of performance depends on how easy it is to access the performance data you collect. This section discusses effective strategies for activities such as managing log files, data archiving, and system analysis to make the data collection process easier, more effective, and more useful.

Controlling Disk Space Used by Log Files

Performance Collection Component provides for automatic management of the log files it creates. You can configure this automatic process or use alternate manual processes for special purposes. The

automatic log file management process works as follows:

Each log file has a configured maximum size. Default maximum sizes are provided when the Performance Collection Component is first installed. However, you can reconfigure these values.

As each log file reaches its maximum size, a “roll back” is performed at mainttime by the scope data collector. During this roll back, the oldest 25 percent of the data in the log file is removed to make room for new data to be added.

Automatic log file maintenance is similar, but not identical, for data collected by scope and by the DSI logging process. For more information on DSI log file maintenance, see [Overview of Data Source Integration](#).

Setting mainttime

Normally, `scope` will only perform log file roll backs at a specific time each day. This is to ensure that the operation is performed at off peak hours and does not impact normal system usage. The time the log files are examined for roll back is set by the `mainttime` parameter in the `parm` file.

Setting the Maximum Log File Size

Choosing a maximum log file size should be a balance between how much disk space is used and how much historical data is available for immediate analysis. Smaller log file sizes save disk space, but limit how much time can be graphed by tools such as Performance Manager. Some ways to reconfigure the scope log file sizes are discussed below.

`scope` logs different types of data into their own log files. This is to allow you to choose how much disk space you want to dedicate to each type independently. For example, global data is fairly compact, but you will often want to go back and graph data for a month at a time. This allows a good statistical base for trending and capacity planning exercises.

Process data can consume more disk space than global data because it is possible to have many interesting processes every minute. Also, the time-value of process data is not as high as for global data. It may be very important to know details about which process was running today and yesterday. You might occasionally need to know which processes were running last week. However, it is unlikely that knowing exactly which processes were run last month would be helpful.

A typical user might decide to keep the following data online:

- Three months of global data for trending purposes

- One month of process data for troubleshooting

- Three months of application data for trending and load balancing

- Two months of device data for disk load balancing

You can edit the `parm` file to set the `size` parameters for each different log file. The sizes are specified in megabytes. For example:

```
SIZE GLOBAL=10.0 PROCESS=30.0 APPLICATION=20.0 DEVICE=5.0
```

The number of megabytes required to hold a given number of days of data can vary by data type, system configuration, and system activity. The best way to determine how big to make the log files on your system is to collect data for a week or so, then use the `utility` program's `resize` command to

change your log file size. The `resize` command scans the log files and determines how much data is being logged each day. It then converts from days to megabytes for you. This function also updates the **parm** file.

Managing Your Resizing Processes

No additional activities are required once automatic log file maintenance is set up. As log files reach their configured maximum sizes, they will automatically be resized by scope.

scope rolls back log files at the mainttime specified in the **parm** file. If you edit the **parm** file and restart scope, the log files will not be rolled to the new sizes until the mainttime occurs. It is important to have scope running at the specified mainttime time or log files may never be rolled back.

Log files may exceed their configured maximum size during the time between maintenance times without causing an immediate roll back.

A log file will never be resized so that it holds less than one full day's data. That means that the log file will be allowed to grow to hold at least one day's worth of data before it is rolled back. Normally this is not an issue, but if you set the **parm** file parameters to collect a large volume of process or application data or set the size to be too small, this can result in a log file significantly exceeding its configured maximum size before it is rolled back.

The scope checks the available disk space on the file system where the log files reside, periodically at intervals specified in the **parm** file for global data collection. If the available disk space falls below one megabyte, scope takes steps to ensure that it does not use any more available space by doing the following:

Immediately performs the log file maintenance without waiting for the regular log file maintenance time. If any log files exceed their maximum sizes (and have more than one day's worth of data in them), they will be rolled back.

If, following the log file maintenance, the available disk space is still not greater than one megabyte, scope writes an appropriate error message to its `status.scope` file and stops collecting data.

Data Archiving

Automatic log file management keeps the latest log file data available for analysis. Data from the raw log files are archived. Process data and global data are logged periodically at intervals specified in the **parm** file. For more information, see [Configure Data Logging Intervals](#). To make room for new data, older data is removed when the log files reach their maximum sizes. If you want to maintain log file data for longer periods of time, you should institute a data archiving process. The exact process you choose depends on your needs. Here are a few possibilities:

Size the raw log files to be very large and let automatic log file maintenance do the rest. This is the easiest archiving method, but it can consume large amounts of disk space after several months.

Extract the data from the raw log files into extracted archive files before it is removed from the raw log files. Formulate a procedure for copying the archive files to long term storage such as tape until needed.

Extract only a subset of the raw log files into extracted archive files. For example, you may not want to archive process data due to its high volume and low time-value.

Some combination of the preceding techniques can be used.

We recommend the following procedures for data archiving:

Size the raw log files to accommodate the amount of detail data you want to keep online.

Once a week, copy the detailed raw data into files that will be moved to offline storage.

Managing Your Archiving Processes

Resize your raw log files as described in the preceding section. Choose log file sizes that will hold at least two weeks worth of data (assuming the archival processing will only be done once a week).

Once a week, schedule a process that runs the **extract** program. The following example shows a script that would perform the weekly processing:

```
#extract -gapdt -xm
```

Each week during the month the data will be appended to the prior week's data. When a new month starts, **extract** creates a new archive log file and splits that week's data into the appropriate monthly archive log file. The log files are named **rxmo** followed by four digits for the year and two more digits for the month. (For example, data for December 1999 would be available in a file named **rxmo199912**.)

At the beginning of each month the previous month's log file is completed and a new log file is started. Therefore, whenever more than one **rxmo** log file is present, it is safe to copy all but the latest one to offline storage until it is needed. When you need to access archived data, restore the desired archival file and access it using the **extract** or **utility** programs.

Depending on your system configuration and activity levels, the amount of disk space accumulated in one month may be large. If this is the case, you can break the detail archive file into smaller files by substituting the weekly command **-xw** in place of **-xm** as shown in the example.

Another alternative is to choose not to archive the detailed process data.

The detailed extraction discussed in the previous example preserves all of your collected performance data. If ever you need to investigate a situation in depth, these files can be restored to disk and analyzed.

Tip: You can use the **extract** program to combine data from multiple extracted files or to make a subset of the data for easier transport and analysis. For example, you can combine data from several yearly extracted files in order to do multiple-year trending analysis.

Note: Moving log files that were created on an HP Operations agent node to a system using an older version of the HP Performance Agent is not supported.

Chapter 3: Working with the HP Operations Agent

After configuring the data collection mechanism, if you want to use the agent in conjunction with HPOM, you can use different components of the Operations Monitoring Component by deploying HPOM policies on the node. For example, if you deploy a measurement threshold policy, the monitor agent component starts operating. Although you can provide most of the monitoring details in the HPOM policies, some components might still require additional configuration to be performed on the node.

Configuring the Monitor Agent

You can start and configure the monitor agent to monitor different sources. When you deploy a measurement threshold policy on a node, the monitor agent component starts operating. Based on the specification in the policies, the agent starts monitoring objects from the following types of sources:

- **External:** An external program that can send numeric values to the agent.
- **Embedded Performance Component:** The data available in the agent's data store.
- **MIB:** Entries in the Management Information Base (MIB).
- **Real Time Performance Management:** Windows performance logs and alerts.
- **Program:** An external program that is started by HPOM and sends numeric values to the agent.
- **WMI:** The WMI database.

To use HPOM policies to monitor the objects from the above sources, see the following topics:

- *For HPOM for Windows:* The *Event Policy Editors* section in the *HPOM for Windows Online Help*.
- *For HPOM on UNIX/Linux:* The *Implementing Message Policies* section in the *HPOM for UNIX 9.10 Concepts Guide*.

Configure the Agent to Monitor MIB Objects

After you deploy the measurement threshold policies (with the Source type set to MIB) on the node, the monitor agent starts querying the MIB objects that can be accessed with the public community string. If you want to configure the monitor agent to use a non-default community string, follow these steps:

1. Log on to the node with the root or administrative privileges.
2. Go to the command prompt (shell).
3. Go to the following directory:

Windows:

`%ovinstalldir%bin`

HP-UX, Solaris, or Linux:

```
/opt/OV/bin
```

AIX:

```
/usr/lpp/OV/bin
```

4. Run the following command:

- To use a non-default community string:

```
ovconfchg -ns eaagt SNMP_COMMUNITY <community_string>
```

In this instance, *<community_string>* is the non-default community string of your choice.

- To use different community strings:

```
ovconfchg -ns eaagt SNMP_COMMUNITY_LIST <list_of_community_strings>
```

In this instance, *<list_of_community_strings>* is a comma-separated list of community strings of your choice. The HP Operations agent processes the list of community strings in the order you specified them with the above command.

For example:

```
ovconfchg -ns eaagt SNMP_COMMUNITY_LIST "C1,C2,C3"
```

The HP Operations agent first tries to establish an SNMP session with the nodes and attempts to perform an SNMP Get operation for the OIDs using the community string C1. If the operation is not successful, the HP Operations agent performs the same operation with the community string C2, and so on.

Note: If the HP Operations agent fails to use all the community strings specified with `SNMP_COMMUNITY_LIST`, it tries to use the community string specified with `SNMP_COMMUNITY`. If the agent fails to get data with all the specified community string, it starts using the default community string `public`.

Persistence of Monitored Object

You can configure the HP Operations agent to periodically store the values of monitored objects and session variables. Storing the values of monitored objects and session variables ensures that the values are preserved and available for use in the event of an interruption or failure.

The `OPC_MON_SAVE_STATE` variable enables you to configure the agent to preserve the values of monitored objects and session variables.

To make sure that the agent is configured to periodically store values of monitored objects and session variables, follow these steps:

1. Log on to the node as root or administrator.
2. Run the following command:

```
ovconfchg -ns eaagt -set OPC_MON_SAVE_STATE TRUE
```

The agent starts preserving the values of monitored objects and session variables.

Installation of the HP Operations agent 11.16 affects the `OPC_MON_SAVE_STATE` variable in the following way:

- If you did not set the variable to any values prior to installing the HP Operations agent 11.16, the `OPC_MON_SAVE_STATE` variable assumes the value `FALSE`.

- If you used the `ovconfchg` command to set a value (TRUE or FALSE) for the variable prior to installing the HP Operations agent 11.16, the configured value remains in effect after the installation process is complete.

For example, if you used the command `ovconfchg -ns eaagt -set OPC_MON_SAVE_STATE TRUE` before installing the version 11.16, the same value (TRUE) is retained after the installation of the HP Operations agent 11.16.

Configuring the Event Interceptor

By default, the event interceptor can collect SNMP traps originating from remote management stations or SNMP-enabled devices, and then can generate appropriate events based on the configuration.

Note: The event interceptor (`opctrapi`) does not format the textual part of MIB. For example, the message text shows the MIB as `.1.3.6.1.4`, and not as `.iso.identified-organization.dod.internet.private`.

You can modify the default behavior of the event interceptor by configuring the following properties:

- `SNMP_TRAP_PORT`: The default port is **162**. You can modify this value to any available port on the HP Operations agent node.
- `SNMP_TRAP_FORWARD_DEST_LIST`: With this property, you can set the address of the remote management station where you want to forward all the available SNMP traps. You can specify multiple system names (with port details) separated by commas.
- `SNMP_TRAP_FORWARD_ENABLE`: By default, this property is set to **FALSE**. By setting this property to **TRUE**, you can enable the event interceptor to forward the SNMP traps available on the HP Operations agent node to remote machines or management stations.
- `SNMP_TRAP_FORWARD_COMMUNITY`: With this property, you can specify the community string of the source machines of the incoming traps and the target machine where you want to forward the SNMP traps. The community strings of the source machines must match with the community strings of the target machines.
- `SNMP_TRAP_FORWARD_FILTER`: With this property, you can filter the available SNMP traps by their OIDs and forward only select traps to the remote machine. The filtering mechanism takes effect with the wildcard (*) character. For example, if you set this property to `1.2.3.*.*`, the event interceptor will forward all the SNMP traps with the OIDs that begin with 1.2.3. By default, all the available traps are forwarded when you enable the event interceptor to forward traps.

Note: If the community string of the source machines do not match with the community string of the target machines, the trap forwarding function fails.

To modify the default behavior of the event interceptor, follow these steps:

1. Log on to the node with the necessary privileges.
 2. In the command prompt, run the following commands:
- To modify the port number, run the following command:

```
ovconfchg -ns eaagt set SNMP_TRAP_PORT <port_number>
```

You must specify an integer value for `<port_number>`. Make sure the specified `<port_number>` is available for use.

- To enable the event interceptor to forward SNMP traps to remote machines, run the following command:

ovconfchg -ns eaagt set SNMP_TRAP_FORWARD_ENABLE TRUE

- If you enable the event interceptor to forward SNMP traps to a remote machine, run the following command to specify the details of the target machine:

ovconfchg -ns eaagt set SNMP_TRAP_FORWARD_DEST_LIST "<machinename>:<port>"

<machinename> is the fully-qualified domain name of the machine where you want to forward the SNMP traps and <port> is the HTTPS port for the machine. If you want to specify multiple targets, separate the machine details with commas.

- If you want to forward only select SNMP traps available on the node to the remote machine, run the following command:

ovconfchg -ns eaagt set SNMP_TRAP_FORWARD_FILTER "<OID Filter>"

<OID Filter> is an OID with the wildcard characters. The event interceptor filters the traps that match the specified OID (with the wildcard characters) from the available traps, and then forwards them to the target machine.

You can configure the event interceptor to use different mechanisms to bind to the port 162 to listen SNMP traps originating from external sources. The SNMP_SESSION_MODE variable enables you to configure this.

- To configure the event interceptor to bind to the port 162 by using the Net-SNMP API to listen to SNMP traps, run the following command:

ovconfchg -ns eaagt set SNMP_SESSION_MODE NETSNMP

- *On Windows nodes only.* To configure the event interceptor to use Microsoft Trap Service to listen to SNMP traps, run the following command:

ovconfchg -ns eaagt set SNMP_SESSION_MODE WIN_SNMP

- *On UNIX/Linux nodes only.* To configure the event interceptor to bind to the port 162 by using the OVSMP API to listen to SNMP traps, run the following command:

ovconfchg -ns eaagt set SNMP_SESSION_MODE NO_TRAPD

- To configure the event interceptor to bind to the port 162 by using the OVSMP API to listen to SNMP traps, run the following command:

ovconfchg -ns eaagt set SNMP_SESSION_MODE NNM_LIBS

- To configure the event interceptor to try to subscribe to the PMD daemon of NNM (7.5x) to listen to SNMP traps, run the following command:

ovconfchg -ns eaagt set SNMP_SESSION_MODE TRY_BOTH

- The event interceptor first tries to subscribe to the PMD daemon of NNM; if the attempt fails, it uses the OVSMP API and binds to the port 162.

- *On UNIX/Linux nodes only.* To configure the event interceptor to subscribe to the PMD daemon of NNM (7.5x) to listen to SNMP traps, run the following command:

ovconfchg -ns eaagt set SNMP_SESSION_MODE NNM_PMD

Message Storm

Message storm occurs when an unusually high number of new messages arrive on the management server within a short time interval and flood the active message browser. This phenomenon can lead to management server outages.

HP Operations agent can detect and suppress the message storm on a managed node. Configure the following properties:

- **OPC_MSG_STORM_DETECTION** - This variable is **mandatory**. By default, this property is set to **FALSE**. By setting this property to **TRUE**, you can enable the message storm detection.
- **OPC_MSG_STORM_DETECTION_CATEGORY** - This variable is **mandatory**. This property is enabled only if the **OPC_MSG_STORM_DETECTION** variable is set to **TRUE**. You can set the variable for any one of the message attributes such as **POLICY**, **MSGGROUP**, **APPLICATION**, **OBJECT**, and **SEVERITY**.
- **OPC_MSG_STORM_RATE** - This variable is **mandatory**. The variable defines the following parameters:
 - **Threshold** - Defines a limit of incoming messages. When the count of incoming messages exceeds the defined limit, message storm condition is detected.
 - **Time** - Interval for which the incoming messages are counted to detect the message storm condition.
 - **Reset** - Defines a limit when the number of messages are below the defined value. This parameter is used to detect when the storm condition is resolved.
- **OPC_SEND_INTERNAL_MSG_ON_MSGSTORM** - This variable is **optional**. This variable defines whether to send or stop the internal messages. By default, the value is set to **TRUE**.
- **OPC_SUPPRESS_MSG_ON_MSG_STORM** - This variable is **optional**. This variable defines whether to send or suppress the messages. The default behavior is that if the threshold condition is met and message storm state is detected, all the messages beyond the threshold value will be suppressed. By default, the value is **TRUE**.
- **OPC_MSG_STORM_TRACE_SUPPRESSED_MSGS** - This variable is **optional**. This variable defines whether to log the messages to the log file only when the **<OPC_SUPPRESS_MSG_ON_MSG_STORM >** is set to **TRUE**. By default, value is **FALSE**.

When message agent detects a message storm or when the message storm gets resolved, a message will be logged into the log file (*System.txt*)-

For Windows: %OvDataDir%\log

For UNIX: /var/opt/OV/log

These parameters are available in the `eaagt.msgstorm` namespace.

The advantages of the detection of the message storm on the managed node are:

- No ECS circuits are required
- Message storm is identified at the source node
- Easy configuration steps and can be configured for various message attributes

Note: Example to detect and suppress the message storm on the managed node.

You can detect the storm condition by setting the `OPC_MSG_STORM_DETECTION` parameter as `TRUE`.

After enabling the message storm condition, define the following parameters -

`OPC_MSG_STORM_DETECTION_CATEGORY` as `POLICY`. Policies deployed are `Opclepolicy` and `Opcmsgipolicy`.

To set the `OPC_MSG_STORM_RATE` parameter, you can calculate the incoming message rate to set the parameters. See "[Message Storm](#)" on the previous page. You can set the values as per the message rate.

Set the values of the `OPC_MSG_STORM_RATE` parameter as `Threshold = 100`, `Time = 20` and `Reset Value = 50`

The `Opclepolicy` sends 50 messages and `Opcmsgipolicy` sends 101 messages. The storm is detected as 101 messages from `Opclepolicy` are more than the set threshold value (100).

You can get the notification when the message storm is detected and resolved by the parameter `OPC_SEND_INTERNAL_MSG_ON_MSGSTORM`. By default, the value is set to `TRUE`.

By default, the message count beyond 100 will be suppressed or ignored. You can set the parameter (`OPC_SUPPRESS_MSG_ON_MSG_STORM`) to `FALSE` to get the messages that are getting suppressed.

You can log the suppressed messages in the log file by setting the parameter (`OPC_MSG_STORM_TRACE_SUPPRESSED_MSGS`).

Configure Message Storm Detection Capabilities

Note: Make sure that you restart the message agent whenever you change any parameter for the configuration. If the message agent is restarted, the content associated with message storm detection will be reset.

To configure the message storm detection or suppression, follow these steps:

1. Log on to the node with the necessary privileges or you can configure remotely.
2. To enable the message storm detection, run the following command:
ovconfchg -ns eaagt.msgstorm -set OPC_MSG_STORM_DETECTION TRUE
The default value is set to `FALSE`.
3. To set the category of the message, run the following command.
ovconfchg -ns eaagt.msgstorm -set OPC_MSG_STORM_DETECTION_CATEGORY <CATEGORY>

Note: This is a mandatory variable.

The category must be defined as **one** of the following:

- POLICY
- GROUP
- APPLICATION
- OBJECT
- SEVERITY

Note: You can only define one of the available options. The combination of various values is not available.

4. Run the command to set the variables of the message rate:

```
ovconfchg -ns eaagt.msgstorm -set OPC_MSG_STORM_  
RATE<ThresholdValue:Time:ResetValue>
```

Note: These variables are mandatory.

The variables are defined as follows:

- **Threshold Value** - Set a numeric value. If the number of messages in the set time interval are more than the threshold value, this condition is known as message storm condition. In this condition, the message agent suppresses the messages till the reset condition is met. The threshold value is checked for the message storm state for all the groups available in the set category. You can check the message rate to set this parameter. See "[Configure Message Storm Detection Capabilities](#)" on the previous page

Example: The category is set to SEVERITY. In this message severity category, there are five types of groups such as Critical, Major, Minor, Warning, and Normal. The messages are grouped based on the severity and are checked for the configured threshold value.

Set the threshold value as 100. The messages received for Critical, Major, Minor, Warning, and Normal severity are 80, 60, 40, 50, and 110 respectively. Here the message storm is detected only for the normal state as the number of messages (110) are more than the set threshold value (100). The remaining 10 messages of the normal severity will be suppressed.

- **Time** - Set the time interval in seconds. The time interval in which the count of incoming messages are recorded. The recommended value is below 900 seconds.
- **Reset Value** - Set a numeric value that must be less than or equal to the threshold value. Message storm detection gets reset if the following conditions are met:
 - Message rate is checked for the set time interval.
 - Message rate should be less than the reset value.

Example to set the parameters for message storm condition.

```
set OPC_MSG_STORM_DETECTION=TRUE
```

```
set OPC_MSG_STORM_DETECTION_CATEGORY=SEVERITY  
set OPC_MSG_STORM_RATE=100:60:45
```

where Threshold value is 100, Time is 60, and Reset value is 45.

After the message storm detection, the number of incoming messages are checked for the set periodic interval (60 seconds). In the set time interval, if the messages are less than the reset value (45), the message agent (opcmsga) stops ignoring the messages and the message storm state is over.

Note: Step 5, 6 and 7 are optional steps.

5. To receive and stop the internal messages, run the following:
ovconfchg -ns eaagt.msgstorm -set OPC_SEND_INTERNAL_MSG_ON_MSGSTORM<Value>
The value must be defined as *one* of the following:
 - TRUE- Internal messages get generated whenever the message storm state is detected and whenever the state is resolved. The default value is TRUE.
 - FALSE- No internal messages are generated for the message storm state and whenever the state is resolved.
6. In the message storm condition, run the following command to suppress or receive the messages:
ovconfchg -ns eaagt.msgstorm -set OPC_SUPPRESS_MSG_ON_MSG_STORM<Value>
The value must be defined as *one* of the following:
 - TRUE- Message agent will suppress messages in the message storm condition. The default value is TRUE.
 - FALSE- Message agent will not suppress the messages in the message storm condition.
7. **Skip this step** if OPC_SUPPRESS_MSG_ON_MSG_STORM variable is set to FALSE.
In the message storm condition, run the following command to log the suppressed messages:
ovconfchg -ns eaagt.msgstorm -set OPC_MSG_STORM_TRACE_SUPPRESSED_MSGS <Value>
The value must be defined as *one* of the following:
 - TRUE- Message agent will save all the suppressed messages in the log file.
 - FALSE- Message agent will not save the suppressed messages. By default, value is FALSE.

The suppressed messages are available in (*msgsuppress.log <process id of the process>*) in the following directory:

For Windows: %OvDataDir%\tmp\OpC
For UNIX: /var/opt/OV/tmp/OpC
8. Run the command to restart the message agent:
ovc -restart opcmsga

Checking the Message Rate

Message rate is used to measure the average rate of messages through the system. After checking the average message rate, you can configure an ideal threshold limit and reset limit to detect and suppress the message storm condition. You can only check the message rate after configuring the message storm detection.

To check the message rate at a particular time, run the following:

- For Windows:
%OVIInstallDir%\bin\eaagt\opcmsga -message_rate
- For UNIX (except AIX):
/opt/OV/bin/eaagt/opcmsga -message_rate
- For AIX:
/usr/lpp/OV/bin/eaagt/opcmsga -message_rate

Integrating HP Operations Agent with HP NNMi

HP Network Node Manager i software (HP NNMi) is a network management software that uses existing SNMP agents for network element discovery and status. Integration with HP Operations agent helps you to monitor the traps forwarded by NNMi and view the enriched SNMP traps on the HPOM console.

The HP Operations agent integration with HP NNMi is available with the following:

- Northbound interface - NNMi northbound interface is used for forwarding NNMi incidents to any application that can receive SNMPv2c traps. SNMP events are sent to SNMP trap interceptor (opctrapi) from NNMi using the northbound interface. To ensure the correct association between the trap-sending device and the event in the receiving application, the rules for these traps must be customized by the varbinds. Integration is done by using the policy which contains the rules for opctrapi to intercept traps forwarded by NNMi northbound interface. Opctrapi will also set various parameters of the outgoing message based on the various configuration such as severity, node name and so on defined in the policy. For more information about using Northbound interface, see *NNMi Deployment Reference, version 9.22*.
- Event forwarding - NNMi SNMP trap forwarding mechanism enriches each SNMP trap before forwarding it to the trap destination. There are two types of event forwarding mechanisms-
 - Original trap forwarding - No additional varbinds are added.
 - Default trap forwarding - NNMi adds varbinds to identify the original source object. This mechanism is used by opctrapi to enhance the SNMP traps.

For more information about the default trap forwarding, see *NNMi help, Configuring Trap Forwarding*.

NNMi forwards the incidents either by NNMi event forwarding or Northbound interface and adds the varbinds to the original event. These varbinds contain additional trap information. To process and use this information with HP Operations agent, you must configure SNMP interceptor (opctrapi) for the following:

- Associate the severity that is available with SNMP trap to the HPOM message. This enhancement will work with Northbound interface only. For more information, see [Integrating with NNMi Northbound Interface to Associate the Message Severity](#).
- Create HPOM Custom Message Attributes (CMA) from NNMi incidents- This enhancement will work with Northbound interface only. For more information, see [Integrating with NNMi Northbound Interface to Add CMA on the HPOM Console](#).
- Derive the source node name where the SNMP trap got originated. This enhancement will work with default trap forwarding only. For more information, see [Integrating with NNMi Trap Forwarding Interface to Assign Source of SNMP Trap](#).

Integrating with NNMi Northbound Interface to Associate the Message Severity

HP Operations agent uses the policy that is generated by the NNMi tool (nrmopcexport.ovpl). This tool (nrmopcexport.ovpl) would set the severity of the SNMP traps. If the severity generated by NNMi tool is Normal, Critical, Warning, Major, or Minor then the message severity would appear as Normal, Critical, Warning, Major, or Minor respectively.

Only when the tool would indicate the severity of the message generated by the HPOM policy as **Unknown**, configure opctrapi to use the available varbind in the trap to derive the value of severity. This configuration helps to associate the severity available with SNMP trap to HPOM message severity.

Configure SNMP Trap Interceptor to Enhance Message Severity

To set the message severity based on severity level available in SNMP trap, you can configure SNMP trap interceptor (opctrapi). Read the severity from the specific OID of SNMP trap and then associate the same severity level to HPOM messages. This configuration step helps opctrapi to use the severity level available in SNMP traps.

Note: The configuration is applicable for the rules where the message severity set in HPOM is Unknown.

You can modify the default behavior of opctrapi by configuring the following properties:

- OPC_SNMP_SET_SEVERITY: By default, this property is set to **FALSE**. By setting this property to **TRUE**, you can enable the event interceptor to read SNMP traps with the specific varbind OID (.1.3.6.1.4.1.11.2.17.19.2.2.12) and set the severity of the message. If this default OID value is not available in SNMP trap, the message severity remains as Unknown.
- OPC_SNMP_OVERRIDE_SEVERITY_OID: You can set the new OID value. The new value will specify the severity of the message.

To configure SNMP interceptor, follow these steps:

Note: You must set the trap interceptor to NETSNMP mode. Run the following command to set the trap interceptor:

```
ovconfchg -ns eaagt set SNMP_SESSION_MODE NETSNMP
```

1. Log on to the node with the necessary privileges.
2. In the command prompt, run the following commands:
 - To enable the SNMP interceptor to read severity levels defined in SNMP traps, run the following command:

```
ovconfchg -ns eaagt.integration.nnm -set OPC_SNMP_SET_SEVERITY TRUE
```

You must specify the boolean value TRUE or FALSE. By default, the value is set to FALSE.

- To set the new OID value, run the following command:

```
ovconfchg -ns eaagt.integration.nnm -set OPC_SNMP_OVERRIDE_SEVERITY_  
OID <OID >
```

<OID > is object identifier. Here, OID is used to derive the severity level of SNMP traps.

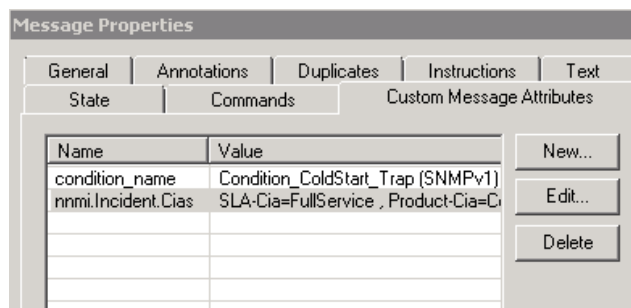
By default, the severity level is derived from the default OID “.1.3.6.1.4.1.11.2.17.19.2.2.12”. You can change the default OID by running the following command:

```
ovconfchg -ns eaagt.integration.nnm -set OPC_SNMP_OVERRIDE_SEVERITY_  
OID <.1.3.6.1.4.1.11.2.17.19.2.2.22>
```

Henceforth, the severity of the message will be based on new OID value <.1.3.6.1.4.1.11.2.17.19.2.2.22>.

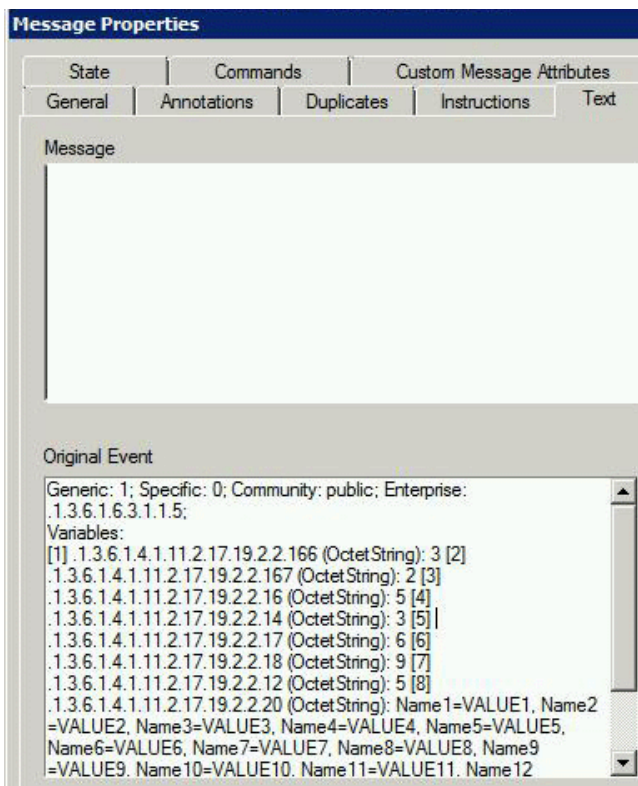
Integrating NNMi Northbound Interface to Add CMA on the HPOM Console

When an SNMP trap is forwarded in NNMi's north-bound integration, the trap includes a varbind that represents the Custom Incident Attributes (CIAs). NNMi CIA appear in a single customer message attribute (CMA) on the HPOM console if CMAs¹ are configured in the HPOM policy. The list appears with CMA name and value pair. This is a default behavior. The messages in CMA tab appear as:



If CMAs are not configured in HPOM policy, NNMi CIA message appears as a text message:

¹A custom message attribute (CMA) can be any information that is meaningful to you and you can have more than one CMA attached to a single message.



After you configure `opctrapi`, the SNMP trap interceptor should access the CIAs and represent each CIA as a Custom Message Attribute (CMA) in the OM message

Configure SNMP Trap Interceptor to Create CMAs from NNMI CIA on the HPOM Console

You can configure `opctrapi` and display NNMI CIA as CMA name and value pair in the CMA properties tab by configuring the following :

- `OPC_SPLIT_NNM_CUSTOM_ATTR`: By default, this property is set to **FALSE**. By setting this property to **TRUE**, all NNMI CIAs values present in the `varbind (.1.3.6.1.4.1.11.2.17.19.2.2.20)` will appear as individual CMA.
- `OPC_SPLIT_NNM_CUSTOM_ATTR_MAX`: This variable is **optional**. This property is enabled only if the `OPC_SPLIT_NNM_CUSTOM_ATTR` variable is set to **TRUE**. The variable defines the number of NNMI custom attributes that can be read and interpreted by HPOM message. By default, the value is set to 20. This means that only 20 CMAs will appear separately with respective values and rest of NNMI CIA will appear in a single CMA. You can specify the value as required.

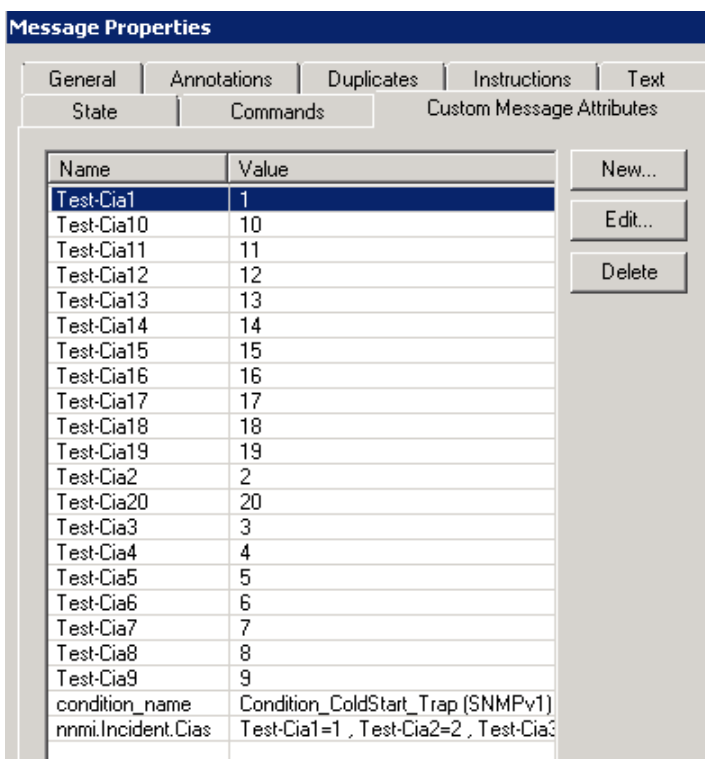
To modify the default behavior of the `opctrapi` to create separate CMA's from NNMI CIAs, follow these steps:

Note: You must set the trap interceptor to NETSNMP mode. Run the following command to set the trap interceptor:

```
ovconfchg -ns eaagt set SNMP_SESSION_MODE NETSNMP
```

1. Log on to the node with the necessary privileges.
2. In the command prompt, run the following commands:
 - To enable opctrapi to read and create a separate CMA from the list that appears in varbind ((.1.3.6.1.4.1.11.2.17.19.2.2.20), run the following command:
ovconfchg -ns eaagt.integration.nnm -set OPC_SPLIT_NNM_CUSTOM_ATTR TRUE
 You must specify a boolean value TRUE or FALSE. By default, the value is set to FALSE.
 - To set the value of the variable to read the message and create separate CMA name and value pair, run the following command:
ovconfchg -ns eaagt.integration.nnm -setOPC_SPLIT_NNM_CUSTOM_ATTR_MAX < Value>
 <Value> is an integer. By default, the value is 20.

After configuration, the messages in CMA tab appear with name and value attributes. In addition the NNMi CIA's untruncated message (nnmi.Incident.Cias) also appears in the list.



Integrating with NNMi Trap Forwarding Interface to Assign Source of SNMP Trap

In the previous versions, whenever NNMi used to forward the SNMP v2c traps to the node (Operations agent is available on the node), the source node name in the HPOM message will appear as NNMi node name.

With this version, HP Operations agent is integrated with NNMi forwarding interface. The NNMi forwarding interface adds varbinds to identify the source where the trap was originated.

You can configure opctrapi to derive the source node that generated the trap. When an HPOM message is generated, the SNMP trap interceptor (opctrapi) should set the source node name as the node where the trap was generated and not NNMi's node name.

Configure SNMP Trap Interceptor to Derive the Source Node Name

You can configure the following property:

OPC_NODENAME_FROM_NNM_FRWD_TRAP: By default, this property is set to **FALSE**. By setting this property to **TRUE**, all the traps are searched for the varbinds “.1.3.6.1.4.1.11.2.17.2.19.1.1.2.0” and .1.3.6.1.4.1.11.2.17.2.19.1.1.3.0. Opctrapi uses the following varbind:

- (.1.3.6.1.4.1.11.2.17.2.19.1.1.2.0) - This varbind is used to identify the IP address type.
- (.1.3.6.1.4.1.11.2.17.2.19.1.1.3.0) - This varbind is used to derive the IP address to set the node as the source node.

To configure opctrapi to assign the source node name when NNMi forwards SNMPv2 events, follow these steps:

Note: You must set the trap interceptor to NETSNMP mode. Run the following command to set the trap interceptor:

```
ovconfchg -ns eaagt set SNMP_SESSION_MODE NETSNMP
```

1. Log on to the node with the necessary privileges.
2. In the command prompt, run the following command to enable opctrapi to read the varbinds and assign the node from where the trap is originating:

```
ovconfchg -ns eaagt.integration.nnm -set OPC_NODENAME_FROM_NNM_FRWD_TRAP TRUE
```

You must specify a boolean value TRUE or FALSE. By default, the value is set to FALSE.

Note: If the variable is not set or set to False, opctrapi will not derive the original source (node name) from where the trap was generated.

Configuring the Backup Server

The message agent sends the messages to the primary server, which is your HP Operations Manager (HPOM). If you have more than one HPOM server in your environment, you can also configure another server to act as the backup server in your environment. When you have a backup server configured in your environment, if the communication link to the primary server is down and the messages cannot be sent to the primary server, the message agent sends the messages to the backup server(s).

You can also configure the backup server for load distribution of the message sync between both the primary and backup servers (for example, in the manager-of-manager (MoM) scenario). For more

information on the MoM scenario, see *HP Operations Manager for UNIX Concepts Guide* or *HP Operations Manager Online Help*. A backup server can be another HPOM server in your environment. You can configure one or more servers as the backup server(s).

You can enable the HP Operations agent to send messages to the backup server by configuring values for the following variables:

- **OPC_BACKUP_MGRS** - You can specify one or a list of servers to be configured as the backup servers. The values in the list can be comma or semicolon separated.
- **OPC_BACKUP_MGRS_FAILOVER_ONLY** - When you set the value for this variable to **TRUE**, the message agent forwards the messages to the backup servers only if the primary server is down. And when the value is set to **FALSE**, messages are sent to the backup servers, irrespective of the status of the primary server. The default value for this variable is **FALSE**.

If you have configured the list of backup servers, during initialization, the message agent creates a list of the backup servers and then checks for the value set for the **OPC_BACKUP_MGRS_FAILOVER_ONLY** variable. When a message arrives at the `msgagtdf` file and the value for the **OPC_BACKUP_MGRS_FAILOVER_ONLY** variable is:

- **FALSE**, the message agent sends the message to the primary server and then to the backup server. After the message is successfully sent to both the servers, the message entries are removed from the `msgagtdf` file.
- **TRUE**, the message agent sends the messages to the primary server. If the message sending fails, message agent tries to forward the message to the backup server. If there are more than one backup servers listed, and the message is delivered to at least one of the backup server, then the message entry is removed from the `msgagtdf` file. Message agent does not try to resend the message to the other backup servers and also does not send the message to the primary server, when it is up again.

Note: When **OPC_BACKUP_MGRS_FAILOVER_ONLY=TRUE**, if a message that started a local automatic action is sent to the primary server, and then the primary server goes down, the message agent sends the action response to backup manager. But this is discarded by the backup manager as the backup server does not have the original message that started the action.

Prerequisites:

- Trusted certificates of the backup servers must be installed on the HP Operations agent node.
- Trusted certificate of the primary server must be installed on the backup server.
- The HP Operations agent node must be added to the backup servers.

To set the values for the variables, follow these steps:

1. Log on to the node as root or administrator.
 2. Run the following commands:
 - To set the backup server, run the following command:
ovconfchg -ns eaagt -set OPC_BACKUP_MGRS <servername>
<servername> is the name of the backup server. For example, *abc.ind.hp.com*.
- To configure a list of backup servers, use either of the following commands:
- **ovconfchg -ns eaagt -set OPC_BACKUP_MGRS <servername1>,<servername2>,...,<servernameN>**
 - **ovconfchg -ns eaagt -set OPC_BACKUP_MGRS <servername1>;<servername2>;...;<servernameN>**

The sequence of the backup servers depend on the sequence in which you specify the server names. In the preceding examples, **<servername1>** acts as the first backup server and **<servername2>** as the second backup server.

- To modify the value for the OPC_BACKUP_MGRS_FAILOVER_ONLY variable, run the following command:

```
ovconfchg -ns eaagt -set OPC_BACKUP_MGRS_FAILOVER_ONLY TRUE
```

The default value is **FALSE**.

Configuring the RTMA Component

The Real-Time Metric Access (RTMA) component provides you with real-time access to system performance metrics, locally or remotely. The perfd process, which is a part of the RTMA component, starts running on the node with the default configuration after you install the HP Operations agent.

Checking the License for Perfd Process

The perfd process starts the data collection only after verifying that the license is enabled. If the license is not enabled, the process is idle. All the activities are recorded in the log files.

Modifying the Settings

You can modify the configuration settings of the perfd process from the **perfd.ini** file, which is available into the following directory on the node:

On Windows: %ovdatadir%

On UNIX (and Linux): /var/opt/perf

Parameter	Description	Default Value
interval	The frequency of data collection in seconds. This value must be a multiple or factor of 60.	10
port	The port used by perfd.	5227
depth	The time duration for which global metric values are retained in the perfd cache. This data is used for data summarization.	30
maxrps	The maximum number of session requests per second accepted by perfd. If the number of requests exceeds the limit, perfd pauses for one second, and then logs the details of this event in the log file. The log file, status-perfd.<port>, is located into the following	20

, continued

Parameter	Description	Default Value
	directory on the node: <i>On Windows: %ovdatadir%</i> <i>On UNIX (and Linux): /var/opt/perf</i>	
maxtpc	The maximum number of sessions per client system accepted by perfd. After the available number of sessions reaches this limit, if an additional request arrives, perfd denies the additional request.	30
maxcps	The maximum number of simultaneous session requests accepted by perfd at a given instant. If the number of requests exceeds the limit, the server will pause for 3 seconds before establishing the sessions.	2
lightweight	If this is set to true, perfd stops collecting data for processes, application, NFS operations, logical systems, and ARM. In addition, the HBA and LVM data on HP-UX will not be collected.	false
localonly	If this is set to true, perfd can be configured only on the local machine. If set to true, perfd denies all connection requests except those coming from the host system (localhost) through the loopback interface. Details of the denied connection requests are logged in the status file.	false
IPv4	This option enables perfd to accept only IPv6 connections. By default, if perfd cannot create an IPv6 socket, it automatically switches to the IPv4-only socket. <div style="background-color: #e0e0e0; padding: 5px;"> Note: If you explicitly want perfd to accept only IPv4 connections, set IPv4 value to true in perfd.ini file. </div>	false

To change the default settings, follow these steps:

On the node, open the **perfd.ini** file with a text editor.

Modify the settings.

Save the file.

Restart the HP Operations agent for the changes to take effect.

Monitoring HP Operations agent on IPv6 connections

On installation, HP Operations agent 11.13 determines the supported IP configuration and binds with the corresponding IP address. Additional configuration is not required for IPv6 connections. For a dual stack node, the node determines the supported server's IP address.

Server	Dual stack node
IPv4	uses IPv4
IPv6	uses IPv6
Dual stack	uses IPv6

Note: IPv6 communication is preferred if both server and node are dual stack.

Restrict Access

You can configure the HP Operations agent to prevent other systems from accessing the real-time performance data of the local system by using the RTMA component.

The `cpsh`, `padv`, and `mpadv` utilities enable you to access the real-time performance data of the agent node remotely. You can use the `cpsh` utility to view the real-time performance data on any remote system where the `perfd` process is running.

You can use the `padv` or `mpadv` utility to run adviser scripts on any remote system where the `perfd` process is running.

These utilities depend on the `perfd` process to access the real-time performance data.

To prevent other systems from accessing the real-time performance data of the local system by revoking their access to the `perfd` process, follow these steps:

Note: This procedure does not prevent the Diagnostic View of HP Performance Manager from accessing the real-time performance data from the system.

1. Log on to the system as an administrator or root. Go to the following directory:

- *On Windows*
`%ovdatadir%`
- *On UNIX/Linux*
`/var/opt/perf`

2. All other systems in the environment (where the HP Operations agent is available) now cannot access the real-time performance data from this system.

3. Create an empty file in this location and save the file as **authip**.

4. To enable a system to access the real-time performance data from this system, open the **authip** file with a text editor, add the FQDN or IP address of the system, and then save the file. You can specify multiple FQDNs or IP addresses (entries must be separated by new lines).
5. Systems that are specified in the **authip** file can now access the real-time performance data from this system.
6. To allow access for all systems, delete the **authip** file.

Configuring the Agent User

The HP Operations agent, after installation, starts running with the Local System account on Windows nodes and with the root account on the UNIX/Linux nodes. You can, however, configure the HP Operations agent to run with a non-default user that has fewer privileges than the root or Local System user.

You can run only the Operations Monitoring Component with a non-root/non-Local System user and the remaining components with the default root/Local System user.

Depending on which user account is used, you can configure the following modes of operation of the agent:

- **Non-privileged:** All components of the HP Operations agent run with a non-default user account that has fewer privileges than root or Local System.

Note: You cannot run the HP Operations Agent in the non-privileged mode on HP-UX.

You must not run HP Operations agent in a non-privileged user mode when both the HP Operations Agent and NNMi coexist on a system.

- **Root:** All components of the HP Operations agent run with the default user (root or Local System). This is the default mode of operation of the agent.
- **Mixed:** The Performance Collection Component runs with the root user and the Operations Monitoring Component runs with the local user account.

Note: While running in the mixed mode, it is recommended that the root or privileged user start the agent processes.

When the agent is configured to run under a non-default user on a system where HP Performance Manager is installed, the `OvTomCatB` service of HP Performance Manager starts running under the non-default agent user.

In an HPOM-managed environment, you can additionally configure the agent perform automatic or operator-initiated commands with a user different from the user it runs under.

Requirements for Using a Non-Default User

The non-default agent user that you want to use must satisfy the following requirements:

- **Windows-only requirements:**
 - The user must have full control of the registry key HKEY_LOCAL_MACHINE/Software/Hewlett-Packard/OpenView
 - The user must have read access to the registry key HKEY_LOCAL_MACHINE/Software/Microsoft/WindowsNT/CurrentVersion/Perflib
 - The user must have rights to:
 - Log on as service — To start and stop the Performance Collection Component services
 - Manage auditing and security logs — The user can view audited events in the security log of the event viewer. A user with this privilege can view and clear the security log.
 - Act as part of the operating system — To control the agent processes.
 - Replace a process-level token — To create the agent processes as the non-root user.
- To monitor a log file using a policy, the agent user must have permission to read that log file.

Note: On Windows, the agent user cannot read the log file since the user does not have permission.

- To start a program using an automatic command, operator-initiated command, tool, or scheduled task, the agent user must have permission to start that program.

On UNIX and Linux systems, if automatic, operator initiated or scheduled action needs to be started with root privileges you must configure the commands using sudo.

You must prefix sudo to the automatic or operator-initiated command as shown in the following example:

```
sudo hostname
```

Note: The non-root users with sudo permission will have root privileges to run the automatic, operator-initiated and scheduled actions.

- Some Smart Plug-ins may require additional configuration or user rights when the agent runs under an alternative user. For more details, see the documentation for individual Smart Plug-ins.

Limitations of Using a Non-Default User

The non-default (non-root or non-privileged) agent user that you want to use has the following limitations:

- The non-privileged user mode is not supported on HP-UX platforms.
- The non-privileged user mode is not supported on the HPOM management server but mixed user mode is supported on the HP Operations Manager for Unix, Linux and Solaris from version 9.20.
- The non-privileged user and mixed modes are not supported on AIX WPAR.
- The BYLS metrics data cannot be collected for Xen and KVM technologies.
- By default, the agent user with non-privileged and mixed user modes will not have permission to read the monitored log file.
- By default, the agent user with non-privileged and mixed user modes will not have permission to start a program using an automatic command, operator-initiated command, tool, or scheduled task.

- HP Operations Smart Plug-ins may require additional configuration or user rights if the agent user with non-privileged and mixed modes does not have administrative rights.
- The HP Operations Agent cannot collect metrics starting with PROC_REGION_* or PROC_FILE_* for all instances of processes owned by non-privileged users. Also, depending on the operating system, processes running with higher privileges such as ovbbccb and sshd will be either available or unavailable in the non-privileged mode.
- On Windows, the metric PROC_USER_NAME is displayed as Unknown for processes owned by users other than the agent user.
- On AIX, you may see the following error message in the command line console (or in the oainstall.log file in the /var/opt/OV/log directory) after you configure the agent to use the non-default user:

```
Product activation failure. Refer to the log file for more details.  
Ignore this error.
```
- On Solaris, Operations Agent gets the process details only up to 80 characters, which is a limitation of Solaris. opcmoma reads the /proc/pid/psinfo file and stores the results in a structure. If extended information is required later, the system reads /proc/pid/as file. If the agent is running as a non-root user and does not have permission to open /proc/pid/as file, then it compares the process details with the limited information available in psinfo.

Configure the Agent User During Installation

At the time of installation, you can configure the HP Operations agent to run under a non-default user (other than root or Local System) on the system. For this purpose, you must install the agent with the help of the profile file for manual installation or the defaults file for HPOM-assisted remote installation. If you cannot configure this at the time of installation, perform the post-installation configuration steps to change the default agent user (see "[Configure the Agent User After Installation](#)" on page 88).

Note: You cannot use this procedure if you want to install the agent on Windows nodes remotely from the HPOM console. While installing the agent on Windows nodes from the HPOM console, install the agent in the *inactive* mode, and then use one of the post-installation configuration procedures to configure the agent to run with a non-default user. For more information, see the *HP Operations Agent and HP Operations Smart Plug-ins for Infrastructure Installation Guide*

To configure the agent during installation to run under a non-default user, follow these steps:

1. Make sure the user is created on the system and the user meets all requirements.
2. If you want to install the HP Operations agent manually on the node, create a **profile** file.
 - a. On the system where you want to install the agent, create a new file and open the file with a text editor.
 - b. Type one of the following statement to specify the mode of agent's mode of operation:
To run the agent under a non-root or non-Local System account, type:
set eaagt:MODE= NPU
To run only the Operations Monitoring Component with a non-root or non-Local System account, type:

set eaagt:MODE= MIXED

- c. Type the following statements:

set eaagt:OPC_RPC_ONLY=TRUE

- d. If you have selected the mixed mode (that is, if you typed `set eaagt:MODE=MIXED` in [step b](#)), type the following statement:

set eaagt:NPU_TASK_SET=EVENT_ACTION

- e. *On Windows only.* Type the following statement:

set eaagt:OPC_PROC_ALWAYS_INTERACTIVE=NEVER

Note: This step is required to successfully run automatic and operator-initiated actions on the node from HPOM when the agent runs in the NPU or MIXED mode.

- f. Type the following statement:

Note: This is a mandatory step for UNIX/Linux nodes. You can skip this step for Windows nodes, but it is recommended that you configure these settings for Windows nodes as well.

set bbc.cb:SERVER_PORT=<Comm_Port>

set eaagt:SNMP_TRAP_PORT=<SNMP_Port>

Note: Since the default communication port for the agent is 383 and the non-root user on UNIX/Linux does not have the permission to access ports below 1024, you must perform this step to assign a non-default communication port to the agent.

In this instance,

<Comm_Port> is the communication port number of your choice.

<SNMP_Port> is the port at which the HP Operations agent receives SNMP traps.

These ports must be higher than 1024 since a non-root user on UNIX/Linux cannot access ports below 1024.

- g. Type the following statements:

set ctrl.sudo:OV_SUDO_USER=<User_Name>

set ctrl.sudo:OV_SUDO_GROUP=<User_Group>

In this instance, *<User_Name>* is the name of the non-default user; *<User_Group>* is the group where the non-default user belongs.

If you want to install the agent remotely from the HPOM console, configure installation defaults:

Note: You cannot use this procedure if you want to install the agent on Windows nodes. For Windows nodes, configure the agent user by manually installing the agent on the node or by performing the post-installation configuration.

- a. Go to the following directory:

On HPOM for Windows

<share_dir>\conf\PMAD

On HPOM on UNIX/Linux

```
/etc/opt/OV/share/conf/OpC/mgmt_sv
```

- b. *For HPOM for Windows.* Save the `agent_install_defaults.cfg.sample` file as `agent_install_defaults.cfg`.
- c. *For HPOM on UNIX/Linux.* Save the `bbc_inst_defaults.sample` file as the `bbc_inst_defaults` file.
- d. Open the file with a text editor.
- e. Add the following content:

```
[eaagt]
<node_details> : MODE=<MODE>
<node_details> : OPC_RPC_ONLY=TRUE
<node_details>: NPU_TASK_SET=EVENT_ACTION
<node_details>: SNMP_TRAP_PORT=<SNMP_Port>
[ctrl.sudo]
<node_details> : OV_SUDO_USER=<User_Name>
<node_details> : OV_SUDO_GROUP=<User_Group>
[bbc.cb]
<node_details>: SERVER_PORT=<Comm_Port>
```

In this instance:

`<node_details>` is a pattern that matches one or more node names or IP addresses. Use standard HPOM pattern syntax. For example:

- `node1.example.com` matches any node with a name that contains the string `node1.example.com`
- `example.com$` matches any node with a name that ends with `example.com`
- `^192.168.<<#> -lt 10` matches any node with an IP address in the range `192.168.0.0` to `192.168.9.255`

`<MODE>` is the mode of operation of the agent (NPU or MIXED)

`<User_Name>` is the name of the non-default user

`<User_Group>` is the group where the non-default user belongs.

`<Comm_Port>` is the communication port number of your choice. This must be higher than 1024. You must also configure the management server so that it connects to `<Comm_Port>` when it communicates with this node. For more information, see “Configuring the Communication Broker Port” on *HP Operations Agent and HP Operations Smart Plug-ins for Infrastructure Installation Guide 11.12*

`<SNMP_Port>` is the port at which the HP Operations agent receives SNMP traps. This must be higher than 1024 so that all the SNMP traps from various sources can be sent to the `<SNMP_Port>`.

- f. Save the file.
3. Install the agent. See the *HP Operations Agent and HP Operations Smart Plug-ins for Infrastructure Installation Guide* for more information about agent installation.

Configure the Agent User After Installation

If you are not able to configure the HP Operations agent to use a non-default user at the time of installation, you can use the `-configure` option of the `oainstall` script (which is available on the agent node) or the `ovswitchuser` command after installation to complete this configuration.

Change the Default User on Windows

If you are not able to configure the agent to run with a non-default user at the time of installation (see ["Configure the Agent User During Installation" on page 85](#)), it is recommended that you install the in the *inactive* mode. For more information, see the *HP Operations Agent Interactive Installation Guide*.

You can use one of the following methods to configure the agent to use a non-default user:

- [Use a Profile File](#)
- [Use the `ovswitchuser` Command](#)

Alternative Method: Use the `ovswitchuser` Command

Note: Make sure to stop all the Operations agent processes before you start using `ovswitchuser.vbs` command.

If you do not want to use a profile file, follow these steps:

Note: Configuration with the profile file is the recommended configuration procedure.

1. Stop the agent:

On Windows 64-bit nodes

```
%ovinstalldir%bin\win64\opcagt -kill
```

On other Windows nodes

```
%ovinstalldir%bin\opcagt -kill
```

2. Run the following command to configure the agent to run with a non-default user:

```
cscript "%ovinstalldir%bin\ovswitchuser.vbs" -existinguser<DOMAIN\USER>-existinggroup<GROUP>-passwd<PASSWORD>
```

In this instance:

<DOMAIN\USER> is the domain and user name.

<GROUP> is the name of the group that the user belongs to, for example AgentGroup.

<PASSWORD> is the user's password.

Note: The command assigns the user rights required for basic agent functionality at group level, not to the individual user. Therefore, take care when you select the group to use. It is advisable to create a new group specifically for the agent user, and add the agent user as a member.

3. Run the following command to set necessary permissions to the non-default user:
cscript %ovinstallDir%\bin\xpl\ovsetscmppermissions.vbs -user <User_Name> -f
In this instance, <User_Name> is the name of the non-default user.
4. Run one of the following commands:
To run all components of the agent under a non-Local System account:
On Windows 64-bit nodes
%OvInstallDir%\bin\win64\lovconfchg -ns eaagt -set MODE NPU
On all other Windows nodes
%OvInstallDir%\bin\lovconfchg -ns eaagt -set MODE NPU
To run only the Operations Monitoring Component with a non-Local System account:
On Windows 64-bit nodes
%OvInstallDir%\bin\win64\lovconfchg -ns eaagt -set MODE MIXED
On all other Windows nodes
%OvInstallDir%\bin\lovconfchg -ns eaagt -set MODE MIXED
5. Run the following commands:
On Windows 64-bit nodes
%OvInstallDir%\bin\win64\lovconfchg -ns ctrl.sudo -set OV_SUDO_USER<User_Name>
%OvInstallDir%\bin\win64\lovconfchg -ns ctrl.sudo -set OV_SUDO_GROUP<Group_Name>
%OvInstallDir%\bin\win64\lovconfchg -ns eaagt -set OPC_PROC_ALWAYS_INTERACTIVE NEVER
On all other Windows nodes
%OvInstallDir%\bin\lovconfchg -ns ctrl.sudo -set OV_SUDO_USER<User_Name>
%OvInstallDir%\bin\lovconfchg -ns ctrl.sudo -set OV_SUDO_GROUP<Group_Name>
%OvInstallDir%\bin\lovconfchg -ns eaagt -set OPC_PROC_ALWAYS_INTERACTIVE NEVER
In this instance, <User_Name> is the name of the non-default user; <Group_Name> is the group where the non-default user belongs.
6. If you have chosen the non-privileged mode of operation, run the following command:
On Windows 64-bit nodes
%OvInstallDir%\bin\win64\lovconfchg -ns eaagt -set OPC_RPC_ONLY TRUE
On all other Windows nodes
%OvInstallDir%\bin\lovconfchg -ns eaagt -set OPC_RPC_ONLY TRUE
7. If you have chosen the mixed mode of operation, run the following command:
On Windows 64-bit nodes
%OvInstallDir%\bin\win64\lovconfchg -ns eaagt -set NPU_TASK_SET EVENT_ACTION
On all other Windows nodes
%OvInstallDir%\bin\lovconfchg -ns eaagt -set NPU_TASK_SET EVENT_ACTION
8. Run the following command:

Note: This is a requirement for UNIX/Linux nodes and not a mandatory step on Windows nodes. However, it is recommended that you complete this step on Windows nodes as well.

On Windows 64-bit nodes

```
%OvInstallDir%bin\win64\ovconfchg -ns eaagt -set SNMP_TRAP_PORT <SNMP_Port>
```

```
%OvInstallDir%bin\win64\ovconfchg -ns bbc.cb -set SERVER_PORT <Comm_Port>
```

On all other Windows nodes

```
%OvInstallDir%bin\ovconfchg -ns eaagt -set SNMP_TRAP_PORT <SNMP_Port>
```

```
%OvInstallDir%bin\ovconfchg -ns bbc.cb -set SERVER_PORT <Comm_Port>
```

In this instance,

<Comm_Port> is the communication port number of your choice. If you set SERVER_PORT to 383, no additional configuration is required. If you do not set the port value to 383, make sure to configure the port on the HPOM management server.

<SNMP_Port> is the port at which the HP Operations agent receives SNMP traps.

These ports must be higher than 1024.

9. Restart the agent:

On Windows 64-bit nodes

```
%ovinstalldir%bin\win64\opcagt -start
```

On all other Windows nodes

```
%ovinstalldir%bin\opcagt -start
```

Change the Default User on UNIX/Linux

If you are not able to configure the agent to run with a non-default user at the time of installation (see ["Configure the Agent User During Installation" on page 85](#)), it is recommended that you install the in the *inactive* mode. For more information, see the *HP Operations Agent Interactive Installation Guide*.

You can use one of the following methods to configure the agent to use a non-default user:

- [Use a Profile File](#)
- [Use the ovswitchuser Command](#)

Use a Profile File

To change the default agent user with the help of a profile file, follow these steps:

1. On the system where you want to install the agent, create a new file and open the file with a text editor.
2. Type one of the following statement to specify the mode of agent's mode of operation:

To run the agent under a non-Local System account, type:

```
set eaagt:MODE=NPU
```

To run only the Operations Monitoring Component with a non-Local System account, type:

```
set eaagt:MODE=MIXED
```

3. If you have selected the non-privileged mode (that is, if you typed `set eaagt:MODE=NPU` in the previous step), type the following statement:

```
set eaagt:OPC_RPC_ONLY=TRUE
```

4. If you have selected the non-privileged mode (that is, if you typed `set eaagt:MODE=NPU` in the previous step), type the following statements:

```
set eaagt:SNMP_TRAP_PORT=<SNMP_port_number>
```

```
set bbc.cb:SERVER_PORT=<Comm_port_number>
```

Note: This is a requirement for UNIX/Linux nodes since the non-root user on UNIX/Linux does not have the permission to access ports below 1024.

In this instance,

`<Comm_Port>` is the communication port number of your choice.

`<SNMP_Port>` is the port at which the HP Operations agent receives SNMP traps.

These ports must be higher than 1024.

5. If you have selected the mixed mode (that is, if you typed `set eaagt:MODE=MIXED` in [step 2](#)), type the following statement:

```
set eaagt:NPU_TASK_SET=EVENT_ACTION
```

6. Type the following statements:

```
set ctrl.sudo:OV_SUDO_USER=<User_Name>
```

```
set ctrl.sudo:OV_SUDO_GROUP=<Group_Name>
```

In this instance, `<User_Name>` is the name of the non-default user; `<Group_Name>` is the group where the non-default user belongs.

7. Save the file into a local directory on the system.
8. Reconfigure the agent to run with the user specified in the profile file:
 - a. Go to the following location on the node:

```
/opt/OV/bin/OpC/install
```

- b. Run the following command:

```
./oainstall.sh -a -configure -agent_profile<path>/<profile_file>
```

In this instance, `<profile_file>` is the name of the profile file; `<path>` is the complete path to the profile file.

Alternative Method: Use the ovswitchuser Command

Note: Make sure to stop all the Operations agent processes before you start using `ovswitchuser.sh` command.

If you do not want to use a profile file, follow these steps:

Note: Configuring with the profile file is the recommended configuration procedure.

1. Go to the following directory:
On HP-UX, Linux, and Solaris

/opt/OV/bin

On AIX

/usr/lpp/OV/bin

2. Stop the agent:

./opcagt -kill

3. Run the following command to configure the agent to run with a non-default user:

./ovswitchuser.sh -existinguser<User_Name>-existinggroup<Group_Name>

In this instance:

<User_Name> is the name of the user that the agent runs under.

<Group_Name> is the name of the group that the user belongs to, for example AgentGroup. The command gives this group full control of all files in the agent data directory, and also full control of all installed packages. If you previously started the command and specified a different group, the command removes control of the files for the previous group.

The group ID flag is set on the agent's data directories. This flag means that the group that you specify will also own any new files and subdirectories in the agent's base directories.

Note: The command assigns the user rights required for basic agent functionality at group level, not to the individual user. Therefore, take care when you select the group to use. It is advisable to create a new group specifically for the agent user, and add the agent user as a member.

4. Run one of the following commands:

To run all components of the agent under a non-root user:

./ovconfchg -ns eaagt -set MODE NPU

To run only the Operations Monitoring Component with a non-root user:

./ovconfchg -ns eaagt -set MODE MIXED

5. Run the following command:

Note: This is a requirement for UNIX/Linux nodes since the non-root user on UNIX/Linux does not have the permission to access ports below 1024.

./ovconfchg -ns eaagt -set SNMP_TRAP_PORT <SNMP_Port>

./ovconfchg -ns bbc.cb -set SERVER_PORT<Comm_Port>

In this instance,

<Comm_Port> is the communication port number of your choice. If you set SERVER_PORT to 383, no additional configuration is required. If you do not set the port value to 383, make sure to configure the port on the HPOM management server.

<SNMP_Port> is the port at which the HP Operations agent receives SNMP traps.

These ports must be higher than 1024.

6. Run the following commands to start the agent:

./opcagt -start

After you configure the agent to run as a non-root user, the following error message may appear in the **System.txt** file:

ovbbccb (22461/1): (bbc-188) Cannot change the root directory for the current process.

Ignore this error.

Change the Default User for Commands

By default, the agent starts automatic or operator-initiated commands under the user account that the agent itself is currently running under. However, you can configure an HP Operations agent to start commands under a different user account. You do this by setting the OVO_STD_USER parameter in the eaagt name space on the nodes. You can configure this parameter in the following ways:

- Configure the values in the HP Operations agent installation defaults. This is recommended if you need to configure the user for large numbers of nodes. You must plan and configure the installation defaults before you create or migrate your nodes.
- Use ovconfchg or ovconfpar at a command prompt.
- Specify the value of OVO_STD_USER in the format `<user>/|<encrypted password>`

Replace `<user>` with the name of the user. For a domain user, specify the domain and user name, for example, EXAMPLE\AgentUser. For a local user, specify just the name, for example AgentUser.

Replace `<encrypted password>` with output from the command `opcpwcrpt <password>`. You can start this command from a command prompt on the management server.

On Windows:

Run as `$OVO_STD_USER`

On HP-UX/Linux/Solaris:

`$OVO_STD_USER` followed by action

Note: OVO_STD_USER variable is currently available only for the Windows platform.

It is also possible to use the OVO_STD_USER when you configure or launch a tool. Specify the user name `$OVO_STD_USER` and leave the password blank.

You must test whether the user account has appropriate rights to run commands and tools correctly.

Note: If the agent fails to start a command or tool as the OVO_STD_USER, the agent may start the command or tool under the same user account that the agent is currently running under. This can happen, for example, if you specify an incorrect user or password.

Configuring the Security Component Variables

RSA certificates and asymmetric encryption are used for secure communication during the SSL handshake between the nodes and between the node and management server. When you install Operations Agent, the Certificate Management component, OvSecCm, generates the RSA key pair used for secure communication, based on the asymmetric key length value configured. If you increase the key length, it increases the strength of the encryption.

The configurable asymmetric key length feature helps users to configure the RSA key length to a required value. By default, the value is set to 2048 and OvSecCm creates 2048 bit RSA key pair for

communication. CA certificates are created with the alias <CA_ovcoreid_keylength>. For example, CA_8cd78962-ab51-755c-1279-85f5ba286e97_2048, where 2048 is the default key length. The configuration variable provided to change the RSA key length is **ASYMMETRIC_KEY_LENGTH**.

When you install HP Operations agent, the Certificate Client component, OvSecCc, sets the configuration variable ASYMMETRIC_KEY_LENGTH to 2048.

To change the RSA key length, you must update the variable ASYMMETRIC_KEY_LENGTH.

To apply the configuration changes on the management server, you can use the MigrateAsymKey tool. This tool is used to create a new CA's key pair, add a new CA certificate for the newly generated key pair, update trusted certificates for local agent and all others OV Resource Groups (ovrg's) on the management server, and create a new certificate for the local agent and all ovrg's.

On the management server

Follow these steps to update the configuration variable on the management server:

1. Update the configuration variable ASYMMETRIC_KEY_LENGTH using the following command:

```
ovconfchg -ns sec.cm -set ASYMMETRIC_KEY_LENGTH <RSA Encryption algorithm supported key length>
```

2. Go to the following location on the management server:

- **On Windows**

```
%ovinstalldir%\sbin\seccs\install\
```

- **On UNIX**

```
%ovinstalldir%/sbin/seccs/install/
```

3. Run the migration tool:

- **On Windows**

```
cscript MigrateAsymKey.vbs -createCAcert
```

- **On UNIX**

```
./MigrateAsymKey.sh -createCAcert
```

The command creates a new CA certificate according to the new key length value set in the configuration variable ASYMMETRIC_KEY_LENGTH.

Note: During an upgrade of the Certificate Server component, running the migration tool with -createCAcert option is performed after installation. -createCAcert is used to create CA certificate corresponding to the new configuration value set to variable ASYMMETRIC_KEY_LENGTH and update the CA certificate created for agent on server and all ovrg's. So, do not run the migration tool with the -createCAcert option unless there is any configuration change made to the ASYMMETRIC_KEY_LENGTH configuration variable.

4. To update the trusted certificates on all the nodes managed by the server, run the command:

```
ovcert -updatetrusted
```

Note: You must run the ovcert -updatetrusted command on all the managed nodes before you run MigrateAsymKey tool with --createNodecert option.

5. To create a new node certificate for agent on the management server and all ovrg's on the server, run the command:

On Windows

```
cscript MigrateAsymKey.vbs -createNodecert
```

On UNIX

```
./MigrateAsymKey.sh -createNodecert
```

The command creates the new node certificates for Operations agent on the management server and all ovrg's with the new RSA key pair.

Note: You can verify the updated key length by the running the command: `ovcert -certinfo <certificate_alias>`. The key length is updated with the `ASYMMETRIC_KEY_LENGTH`.

On the agent

To apply the configuration changes on the agent (managed node), follow these steps:

1. Update the configuration variable `ASYMMETRIC_KEY_LENGTH` using the following command:

```
ovconfchg -ns sec.cm -set ASYMMETRIC_KEY_LENGTH <RSA Encryption algorithm supported key Length>
```
2. To remove the existing node certificate on the agent, run the following commands:

```
ovcert -remove <certificate alias>  
ovcert -remove <CA certificate alias>
```
3. To request a new node certificate from the management server, run the following command:

```
ovcert -certreq
```

Configuring the Security Component for Symmetric Key

When you install HP Operations agent, the security components `OvSecore` and `OvJSecCore` will continue to use the older default algorithm values for the encryption and decryption. Both `OvSecCore` and `OvJSecCore` are included in the `HPSharedComponent` packages for Windows from version 11.10.

The supported symmetric algorithms are as follows:

- Blowfish
- DES
- DES3
- AES128
- AES192
- AES256

The two configuration variables are:

- **DEF_SYM_KEY_ALGO**- This variable is used to set the default symmetric key algorithm for encryption. The supported algorithm values are:
 - eBlowfish
 - eDES

- eDES3
 - eAES128
 - eAES192
 - eAES256
 - eDefault – uses AES128 as the default algorithm.
- **ENABLE_DEF_SYM_KEY_ALGO**- This variable is used to enable the use of the default symmetric key algorithm set to DEF_SYM_KEY_ALGO. The supported value is TRUE and for any other value it is considered as not set.
You can use a configuration variable to enable the use of the algorithm specified in the configuration for encryption. You can set the value to TRUE by using the MigrateSymKey tool.
To update the configuration settings on the management server and on the nodes running with operations agent, you can use MigrateSymKey tool. The MigrateSymKey tool sets ENABLE_DEF_SYM_KEY_ALGO configuration variable to TRUE and migrates the KeyStore content based on the algorithm set to the configuration variable DEF_SYM_KEY_ALGO.

Follow these steps to update the configuration on the management server:

Note: Set the configuration variable DEF_SYM_KEY_ALGO to any of the following supported algorithms with the command:

```
ovconfchg -ns sec.core -set DEF_SYM_KEY_ALGO <Supported Algorithm>
```

If the variable is not set, then eAES128 is used as the default algorithm.

1. Go to the following location on the management server or on the node where agent is installed:

- **On Windows**

```
%ovinstalldir%\bin\secco\
```

- **On UNIX**

```
%ovinstalldir%/lbin/secco/
```

2. Run the migration tool:

- On Windows**

```
MigrateSymKey
```

- On UNIX**

```
../MigrateSymKey.sh
```

After running the tool, the KeyStore content is encrypted based on the new algorithm set to DEF_SYM_KEY_ALGO.

Configuring viserver for Monitoring vMA Nodes

Skip this section if you do not want to monitor vMA nodes.

You can configure the viserver daemon by modifying the contents of two configuration files—**viserver.properties** and **VILog4j.xml**. These files are available in **/var/opt/perf**.

viserver.properties

This file contains the following parameters:

[port](#)

[hosts](#)

[instance](#)

[jvmArgs](#)

[log4jInterval](#)

You must restart viserver if you change the settings in the **viserver.properties** file. The new settings are effective only after you restart viserver.

port

The port parameter is the loopback port through which viserver and clients communicate. The port parameter is non-editable; the value of this parameter changes when you restart **viserver**.

hosts

The hosts parameter defines the number of hosts that viserver daemon can support. The default value is 20.

If you have more hosts in your environment, you can edit this parameter to specify your required setting. If the HP Operations agent is not able to collect data for the number of hosts that you specified, you must reduce vifp targets.

instance

The instance parameter defines the number of instances viserver can support. The default value is 200.

If you have more instances in your environment, you can edit this parameter to specify your required setting. If the HP Operations agent is not able to collect data for the number of instances that you specified, you must reduce vifp targets. You can also create additional vMA Nodes to monitor remaining targets.

jvmArgs

The jvmArgs parameter enables you to add jvm arguments and modify jvm as required in your environment.

The default configuration for jvmArgs is as follows:

```
jvmArgs=-Xms512m -Xmx2560m -classpath  
/opt/perf/bin/java/activation.jar\:/opt/perf/bin/java/axis-
```

```
ant.jar\:/opt/perf/bin/java/axis.jar\:/opt/perf/bin/java/commons-discovery-  
0.2.jar\:/opt/perf/bin/java/commons-logging-  
1.0.4.jar\:/opt/perf/bin/java/jaxrpc.jar\:/opt/perf/bin/java/log4j-  
1.2.8.jar\:/opt/perf/bin/java/mailapi.jar\:/opt/perf/bin/java/saa.jar\:/opt/perf/b  
in/java/vim25.jar\:/opt/perf/bin/java/viserver.jar\:/opt/perf/bin/java/wsd14j-  
1.5.1.jar\:/opt/vmware/vma/lib64/vmatargetlib25.jar\:/opt/vmware/vma/lib64/vifplib2  
5.jar com.hp.perfagent.VI daemon
```

log4jInterval

The `log4jInterval` parameter specifies the interval at which viserver checks for changes in **VILog4j.xml** file. The default value is 60000 milliseconds (1 minute). You can modify this value as required.

VILog4j.xml

viserver uses **VILog4j.xml** file, located in `/var/opt/perf`, to log status information in the **status.viserver** file. The **log4j.dtd** file, available in `/var/opt/perf`, defines the template for the **VILog4j.xml** file. You can change the configuration settings in the **VILog4j.xml** file; the changes take effect after a specified time. The value of the specified time is defined in the parameter `log4jInterval`.

Note: There are elements in the XML file that are required for the logging to work correctly. Do not change or delete these elements. Only recommended change is the level of the `com.hp.perfagent` logger.

The XML file consists of the following major elements:

appender

logger

You can change only the following item within the `<logger name="com.hp.perfagent"> ... </logger>` element:

```
<level value = "info"/>
```

This entity determines the level of logging in the **status.viserver** file. You can set value to one of the following non-default settings:

fatal: Use this setting to log minimal information.

debug: Use this setting to log information for debugging.

Note: Use the debug setting only for troubleshooting purposes.

Monitoring Applications and Services Logs on Windows

The Logfile Encapsulator component of the HP Operations agent supports the Event Forwarding Feature of Windows Event Log. That is, events which are forwarded from different machines can be read and it also enables you to monitor Windows event logs. The Windows Event Log policies help you configure the agent to monitor Windows event logs of your choice.

The following versions of Windows provide a new category of event logs—Applications and Services logs:

- Windows Vista
- Windows Server 2008
- Windows Server 2008 R2
- Windows 7

You can monitor these Applications and Services logs with the HP Operations agent 11.16 with appropriately configured Windows Event Log policies.

The HP Operations agent cannot monitor the following types of event logs:

- Events originating from a remote system (collected by using the Event Subscription feature of Windows)
- Saved event logs

The HP Operations agent can monitor events with the following event levels:

- Error
- Information
- Warning
- LOG_ALWAYS
- VERBOSE
- Audit Failure
- Audit Success

The following table shows how eventlog fields are displayed in the message browser.

Table 1 Eventlog to Message Browser Field Correlation

Eventlog Field	Message Browser Field	Comments
Date	Date	The date the event was created on managed node.
Time	Time	The time the event was created on managed node.
Event ID	Message Text	The Event ID will be displayed before any additional message text.

Table 1 Eventlog to Message Browser Field Correlation , continued

Eventlog Field	Message Browser Field	Comments
Source	Application	None
Type <i>error</i> <i>error</i> <i>information</i> <i>warning</i> <i>log_always</i> <i>verbose</i> <i>audit failure</i> <i>audit success</i>	Severity <i>critical</i> <i>error</i> <i>normal</i> <i>warning</i> <i>normal</i> <i>normal</i> <i>warning</i> <i>normal</i>	The mapping of Event log type severity to the Operations Manager message severity.
Category	Object	None
Description	Message Text	All other message text (after the Event ID).
User	Not mapped	Not mapped
Computer	Node	The name of the node as it is known to the management server.
	Msg Group	Empty

Monitor Applications and Services Event Logs from HPOM for Windows

To create a Windows Event Log policy for monitoring an Applications and Services log, follow these steps:

1. Log on to the Windows node where the Windows event log exists.
2. Open the Event Viewer window.
3. In the console tree, select the event. In the details pane, the name of the event log appears (next to the Log Name field).

Log Name:	Microsoft-Windows-Bits-Client/Operational
Source:	Bits-Client
Event ID:	306

Note down the name of the log file as it appears in the details pane.

4. Open the HPOM for Windows console.

5. In the console tree, under Agent Policies Grouped by Type, right-click **Windows Event Log**, and then click **New > policy**.
6. The policy editor for the Windows Event Log policy opens.
7. In the Source tab, type the name of the Windows event log (which you noted down in step 3) in the Event Log Name field.

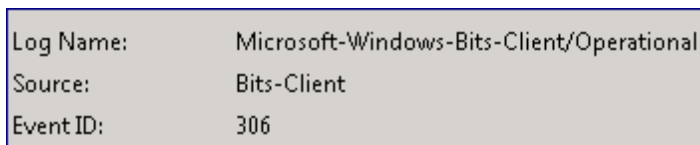


8. Follow the instructions in the HPOM for Windows online help to specify other details in the policy.
9. Save the policy.
10. Deploy the policy on the Windows node.


Monitor Applications and Services Event Logs from HPOM on UNIX/Linux 9.1x

To create a Windows Event Log policy for monitoring an Applications and Services log, follow these steps:

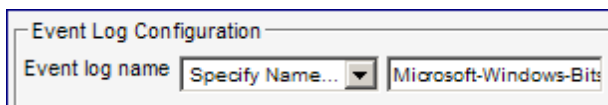
1. Log on to the Windows node where the Windows event log exists.
2. Open the Event Viewer window.
3. In the console tree, select the event. In the details pane, the name of the event log appears (next to the Log Name field).



Note down the name of the log file as it appears in the details pane.

4. Log on to the HPOM for UNIX Administration UI.
5. Click **OMU**.
6. Click **Browse > All Policy Types**.
7. Click Windows Event Log. The Policy Type **Windows_Event_Log** page opens.
8. Click  , and then click **New Policy**. The Add Windows_Event_Log Policy page opens.

In the Source tab, in the Event Log Name field, select Specify Name. A new text box appears. Type the name of the Windows event log (which you noted down in step 3) in the text box.



9. Follow the instructions in the HPOM for UNIX online help to specify other details in the policy.

10. Save the policy.
11. Deploy the policy on the Windows node.

Monitor Applications and Services Event Logs from HPOM for UNIX 8.35

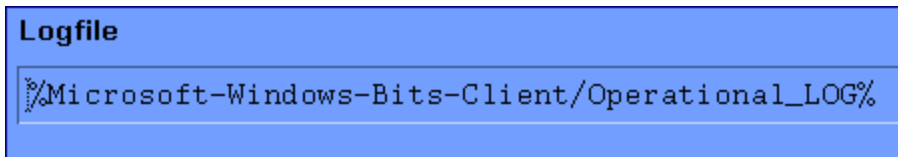
To create a Windows Event Log policy for monitoring an Applications and Services log, follow these steps:

1. Log on to the Windows node where the Windows event log exists.
2. Open the Event Viewer window.
3. In the console tree, select the event. In the details pane, the name of the event log appears (next to the Log Name field).

Log Name:	Microsoft-Windows-Bits-Client/Operational
Source:	Bits-Client
Event ID:	306

Log Name:	Microsoft-Windows-Bits-Client/Operational
Source:	Bits-Client
Event ID:	306

4. Log on to the HPOM for UNIX console.
5. Go to the Message Source Template window and create a new Logfile template.
6. In the Logfile field (in the Add Logfile dialog box), type two % characters, and then, within two % characters, type the name of the Windows event log (which you noted down in step 3) with the suffix `_LOG`. For example, if the event log name is `<event_log_name>`, you must type `%<event_log_name>_LOG%`.



7. Follow the instructions in the HPOM for UNIX online help to specify other details in the policy.
8. Save the policy.
9. Deploy the policy on the Windows node.

Chapter 4: Overview of Node Resolution

A node can have a single or multiple network interfaces. IP address associated with each network interface may have a host name. HP Operations agent selects the host name using operating system specific APIs based on system configuration (example, `gethostname /getaddrinfo /getnameinfo`). It automatically configures `OPC_NODENAME` parameter that provides the value of the local host name used by HP Operations agent. `OPC_NODENAME` is an internal parameter and you cannot configure it. When a node has multiple IP addresses or host names, you can overwrite the default configuration by setting `OPC_IP_ADDRESS` to a specific IP address. The host name associated with the specified IP address is assigned to `OPC_NODENAME`. In case the IP address is not associated to a host name, you can configure `OPC_NAMESRV_LOCAL_NAME` to a specific host name. You can use the `LOCAL_NODE_NAME` variable available under the `xpl.net` namespace to overwrite the local node name of the system. The defined values of `OPC_NODENAME`, `OPC_IP_ADDRESS` and `LOCAL_NODE_NAME` are used to determine several variables in policy. The message variables are as follows:

- `<$MSG_GEN_NODE>`: Returns the IP address of the node that sends the message.
- `<$MSG_GEN_NODE_NAME>`: Returns the host name of the node that sends the message.
- `<$MSG_NODE>`: Returns the IP address of the node on which the original event took place.
- `<$MSG_NODE_NAME>`: Returns the host name of the node on which the original event took place.

Note: The `OPC_NODENAME`, `OPC_IP_ADDRESS` and `OPC_NAMESRV_LOCAL_NAME` parameters are available under the `eaagt` namespace.

Following are some examples:

Example 1:

Following are the IP addresses and host names in a multiple network interface environment:

```
IP1 abc.test.com abc
```

```
IP2 xyz.test.com xyz
```

where IP1 and IP2 are the two different IP addresses.

abc.test.com and xyz.test.com are the Fully Qualified Domain Names (FQDN).

abc and xyz are the host names.

xyz is the host name of the local system.

The two parameters `OPC_NODENAME`, `<$MSG_GEN_NODE_NAME>` automatically set the FQDN value to xyz.test.com and `<$MSG_GEN_NODE>` parameter automatically sets the IP address to IP2. These configurations happen by default because xyz is the host name of the local system.

Choose to configure a different IP address (such as IP1) using the following command:

```
ovconfchg -ns eaagt -set OPC_IP_ADDRESS IP1
```

As a result of this configuration, the parameters `OPC_NODENAME` and `<MSG_GEN_NODE_NAME>` will set the FQDN value to `abc.test.com`. `<MSG_GEN_NODE>` will set the IP address to IP1.

Example 2:

Following are the IP addresses and host names in a multiple network interface environment:

```
IP1 xyz.test.com xyz
```

```
IP2 xyz.test.com xyz
```

where IP1 and IP2 are the two different IP addresses.

`xyz.test.com` is the Fully Qualified Domain Name (FQDN).

`xyz` is the host name of the local system.

The `<MSG_GEN_NODE>` parameter automatically sets the IP address to IP1. This configuration happens by default because `xyz` is the host name of the local system and IP1 is the first IP address (as per the order of IP addresses) that is associated with `xyz`.

Choose to configure a different IP address (such as IP2) using the following command:

```
ovconfchg -ns eaagt -set OPC_IP_ADDRESS IP2
```

As a result of this configuration, `<MSG_GEN_NODE>` will set the IP address to IP2.

Example 3:

In a multiple network interface environment, a Windows IPv6 system has four configured IP addresses in the following order:

```
IP1
```

```
IP2
```

```
IP3
```

```
IP4
```

The IP addresses are associated with host names in the following order; IP1 is not associated to a host name.

```
IP3 xyz.test.com xyz
```

```
IP2 xyz.test.com xyz
```

```
IP4 xyz.test.com xyz
```

By default, the `<MSG_GEN_NODE>` parameter automatically sets the IP address to IP2 because IP2 is the first configured IP address (as per the order of IP addresses) that is associated to the local system name.

Choose to configure a different IP address using the following command:

```
ovconfchg -ns eaagt -set OPC_IP_ADDRESS <IP_address>
```

As a result of this configuration, `<MSG_GEN_NODE>` will set to the specific IP address.

Example 4:

Following are the IP address and host names in a single network interface environment:

```
IP1 abc.test.com abc
```

```
IP1 xyz.test.com xyz
```

where IP1 is the IP address.

abc.test.com and xyz.test.com are the Fully Qualified Domain Names (FQDN).

abc and xyz are the host names.

xyz is the host name of the local system.

By default, the two parameters OPC_NODENAME and <MSG_GEN_NODE_NAME> automatically set the FQDN value to xyz.test.com because xyz is the host name of the local system.

You can choose to configure the IP address IP1 using the following command:

```
ovconfchg -ns eaagt -set OPC_IP_ADDRESS IP1
```

As a result of this configuration, OPC_NODENAME and <MSG_GEN_NODE_NAME> will set the FQDN value to abc.test.com. These configuration happen because abc.test.com is the first FQDN associated with the IP address IP1.

Example 5:

Following are the IP addresses in a multiple network interface environment:

```
IP1
```

```
IP2 xyz.test.com xyz
```

where IP1 and IP2 are the two different IP addresses.

IP1 is not associated to a host name.

xyz.test.com is the Fully Qualified Domain Name (FQDN).

xyz is the host name of the local system.

By default, the two parameters OPC_NODENAME, <MSG_GEN_NODE_NAME> automatically set the FQDN value to xyz.test.com and <MSG_GEN_NODE> parameter automatically sets the IP address to IP2.

Choose to configure a different IP address (such as IP1) using the following command:

```
ovconfchg -ns eaagt -set OPC_IP_ADDRESS IP1
```

As a result of this configuration, OPC_IP_ADDRESS is set to IP1 and is not associated to a host name. You can configure the host name corresponding to OPC_IP_ADDRESS using the below command:

```
ovconfchg -ns eaagt -set OPC_NAMESRV_LOCAL_NAME <host_name>
```

where *<host_name>* is any name like abc.test.com. After running both the commands, the two parameters OPC_NODENAME and <MSG_GEN_NODE_NAME> will set the FQDN value to abc.test.com.

Note: The parameter OPC_IP_ADDRESS set in a NAT environment behaves in the same manner as described in the above examples.

Example 6:

Following are the IP address and host names in a single network interface environment:

```
IP1 abc.test.com abc xyz
```

```
IP1 xyz.test.com xyz
```

where IP1 is the IP address

abc.test.com and xyz.test.com are the Fully Qualified Domain Names (FQDN).

abc and xyz are the host names.

xyz is the host name of the local system.

By default, the two parameters OPC_NODENAME and <MSG_GEN_NODE_NAME> automatically set the FQDN value to abc.test.com. In case the external event originates from node with name xyz then the <MSG_NODE_NAME> parameter automatically sets to xyz.

You can choose to configure OPC_NODENAME, <MSG_GEN_NODE_NAME>, and also <MSG_NODE_NAME> to the FQDN value xyz.test.com by performing one of the following options:

- Interchange the order of the entries in hosts file.
- Remove the alias **xyz** from the first entry.

Example 7:

Following are the IP addresses and host names of two systems:

```
IP1 abc.test.com abc
```

```
IP2 xyz.test.com xyz
```

where IP1 and abc are the IP address and host name of remote system.

IP2 and xyz are the IP address and host name of local system.

abc.test.com and xyz.test.com are the Fully Qualified Domain Names (FQDN).

When an event originates from IP1 (for example, a trap originates from the remote node abc) the following configurations happen by default:

- Parameter <MSG_NODE> sets to IP1 as <MSG_NODE> returns the IP address of the node on which the original event took place.
- Parameter <MSG_NODE_NAME> sets to abc as <MSG_NODE_NAME> returns the host name of the node on which the original event took place.

- Parameter `<$MSG_GEN_NODE>` sets to IP2 as `<$MSG_GEN_NODE>` returns the IP address of the node that sends the message.
- Parameter `<$MSG_GEN_NODE_NAME>` sets to xyz as `<$MSG_GEN_NODE_NAME>` returns the host name of the node that sends the message.

When the `LOCAL_NODE_NAME` variable is configured, the Operations Agent uses `LOCAL_NODE_NAME` as the hostname of the local system.

To configure a different local host name (such as xyz), use the following command:

```
ovconfchg -ns xp1.net -set LOCAL_NODE_NAME xyz
```

As a result of this configuration, the parameters `OPC_NODENAME` and `<$MSG_GEN_NODE_NAME>` will set the `LOCAL_NODE_NAME` variable value to xyz.

Chapter 5: Using the Utility Program

The utility program is a tool for managing and reporting information on log files, the collection parameters (`parm`) file, and the alarm definitions (`alarmdef`) file. You can use the utility program interactively or in batch mode to perform the following tasks.

- Scan raw or extracted log files and produce a report showing:
 - dates and times covered
 - times when the scope collector was not running
 - changes in scope parameter settings
 - changes in system configuration
 - log file disk space
 - effects of application and process settings in the collection parameters (`parm`) file
- Resize raw log files
- Check the `parm` file for syntax warnings or errors
- Check the `alarmdef` file for syntax warnings or errors
- Process log file data against alarm definitions to detect alarm conditions in historical data

This chapter covers the following topics:

- [Running the Utility Program](#)
- [Using Interactive Mode](#)
- [Using the Utility Command Line Interface](#)
- [Utility Scan Report Details](#)

For detailed descriptions about the utility commands see, [Utility Commands](#).

Running the Utility Program

There are three ways to run the utility program:

- Command line mode - You control the utility program using command options and arguments in the command line.
- Interactive mode - You supply interactive commands and parameters while executing the program with `stdin` set to an interactive terminal or workstation.
If you are an experienced user, you can quickly specify only those commands required for a given task. If you are a new user, you may want to use the utility program's `guide` command to get some assistance in using the commands. In guided mode, you are asked to select from a list of options to perform a task. While in guided mode, the interactive commands that accomplish each

task are listed as they are executed, so you can see how they are used. You can quit and re-enter guided mode at any time.

- Batch mode - You can run the program and redirect `stdin` to a file that contains interactive commands and parameters.

The syntax for the command line interface is similar to typical UNIX command line interfaces on other programs and is described in detail in this chapter.

For interactive and batch mode the command syntax is the same. Commands can be entered in any order; if a command has a parameter associated with it, the parameter must be entered immediately after the corresponding command.

There are two types of parameters - required (for which there are no defaults) and optional (for which defaults are provided). How `utility` handles these parameters depends on the mode in which it is running.

- In interactive mode, if an optional parameter is missing, the program displays the default argument and lets you either confirm it or override it. If a required parameter is missing, the program prompts you to enter the argument.
- In batch mode, if an optional parameter is missing, the program uses the default values. If a required parameter is missing, the program terminates.

Errors and missing data are handled differently for interactive mode than for command line and batch mode. You can supply additional data or correct mistakes in interactive mode, but not in command line and batch mode.

Using Interactive Mode

Using the `utility` program's interactive mode requires you to issue a series of commands to execute a specific task.

For example, if you want to check a log file to see if alarm conditions exist in data that was logged during the current day, you issue the following commands after invoking the `utility` program:

```
checkdef /var/opt/perf/alarmdef
detail off
start today-1
analyze
```

The `checkdef` command checks the alarm definitions syntax in the `alarmdef` file and then sets and saves the file name for use with the `analyze` command. The `detail off` command causes the `analyze` command to show only a summary of alarms. The `start today-1` command specifies that only data logged yesterday is to be analyzed. The `analyze` command analyzes the raw log files in the default SCOPE data source against the `alarmdef` file.

Example of Using Interactive and Batch Mode

The following example shows the differences between how the `utility` program's `resize` command works in batch mode and in interactive mode.

The `resize` command lets you set parameters for the following functions:

- Type of log file to be resized.
- Size of the new file.
- Amount of empty space to be left in the file.
- An action specifying whether or not the resize is to be performed.

This example of the `resize` command resizes the global log file so that it contains a maximum of 120 days of data with empty space equal to 45 days. The command and its parameters are:

```
resize global days=120 empty=45 yes
```

The results are the same whether you enter this command interactively or from a batch job.

The first parameter—`global`—indicates the log file to be resized. If you do not supply this parameter, the consequent action for interactive and batch users would be the following:

- Batch users — the batch job would terminate because the `logfile` parameter has no default.
- Interactive users — you would be prompted to choose which type of log file to resize to complete the command.

The last parameter—`yes`—indicates that resizing will be performed unconditionally.

If you do not supply the `yes` parameter, the consequent action for interactive and batch users would be the following:

- Batch users — resizing would continue since `yes` is the default action.
- Interactive users — you would be prompted to supply the action before resizing takes place.

Before using the `resize` command in either batch mode or interactive mode, you must first stop data collection. For details, see [Stopping and Restarting Data Collection](#) in Chapter 2.

Utility Command Line Interface

In addition to the interactive and batch mode command syntax, command options and their associated arguments can be passed to the `utility` program through the command line interface. The command line interface fits into the typical UNIX environment by allowing the `utility` program to be easily invoked by shell scripts and allowing its input and output to be redirected to UNIX pipes.

For example, to use the command line equivalent of the example shown in the previous section "Using Interactive Mode" enter:

```
utility -xr global days=120 empty=45 yes
```

Command line options and arguments are listed in the following table. For more information about the referenced command descriptions see [Utility Commands](#).

Table 2: Command Line Arguments

Command Option	Argument	Description
-b	date time	Specifies the starting date and time of an analyze or scan function. (See start command)

Command Option	Argument		Description
-e	date	time	Specifies the ending date and time of an analyze or scan function. (See stop command)
-l	logfile		Specifies which log file to open. (See logfile command)
-f	listfile		Specifies an output listing file. (See list command)
-D			Enables details for analyze, scan and parm file checking. (See detail command)
-d			Disables details for analyze and parm file for checking. (See detail command)
-v			Echoes command line commands as they are executed.
-xp	parmfile		Syntax checks a parm file. (See parmfile command)
-xc	alarmdef		Syntax checks and sets the alarmdef file name to use with -xa (or analyze command). (See checkdef command)
-xa			Analyzes log files against the alarmdef file. (See analyze command)
-xs	logfile		Scans a log file and produces a report. (See scan command)
-xr	global application process device transaction ls EMPTY=nnn SPACE=nnn	SIZE=nnn DAYS=nnn YES NO MAYBE	Resizes a log file. (See resize command)
-? or ?			Displays command line syntax.

Example of Using the Command Line Interface

The following situation applies when you enter command options and arguments on the command line:

Errors and missing data are handled exactly as in the corresponding batch mode command. That is, missing data is defaulted if possible and all errors cause the program to terminate immediately.

Echoing of commands and command results is disabled. Utility does not read from its `stdin` file. It terminates following the actions in the command line.

```
utility -xp -d -xs
```

Which translates into:

-xp	Syntax checks the default <code>parm</code> file.
-d	Disables details in the scan report.
-xs	Performs the scan operation. No log file was specified so the default log file is scanned.

Utility Scan Report Details

The `utility` program's `scan` command reads a log file and writes a report on its contents. The report's contents depend on the commands issued prior to issuing the `scan` command. (For more information, see the description of the [scan](#) command in [Utility Commands](#))

The following table summarizes the information contained in all scan reports and in reports that are produced only when the `detail on` command is used (the default) with the `scan` command

Table 3: Information Contained in Scan Report

Initial Values	
Initial <code>parm</code> file global information and system configuration information	Printed only if <code>detail on</code> is specified.
Initial <code>parm</code> file application definitions	Printed only if <code>detail on</code> is specified.
Chronological Detail	
<code>parm</code> file global changes	Printed only if <code>detail on</code> is specified.
<code>parm</code> file application changes	Printed only if <code>detail on</code> is specified.
Collector off-time notifications	Printed only if <code>detail on</code> is specified.
Application-specific summary reports	Printed only if <code>detail on</code> is specified.
Summaries	
Process summary report	Always printed if process data was scanned.
Collector coverage summary	Always printed.
Log file contents summary	Always printed. Includes space and dates covered.
Log file empty space summary	Always printed.

Scan Report Information

The information in a utility scan report is divided into three types:

- Initial values
- Chronological details
- Summaries

Initial Values

This section describes the following initial values:

- Initial parm file global information
- Initial parm file application definitions

Initial Parm File Global Information

To obtain this report, use the `scan` command with its default `detail on`.

This report lists the configuration settings of the `parm` file at the time of the earliest global record in the log file. Later global information change notifications are based on the values in this report. If no change notification exists for a particular parameter, it means that the parameter kept its original setting for the duration of the scan.

The following example shows a portion of a report listing the contents of the `parm` file.

```
06/03/99 15:28 System ID="Homer"
scopeux/UX A.10.00 SAMPLE INTERVAL = 300,300,60 Seconds, Log version=D
Configuration: 9000/855, O/S A.10.00 CPUs=1
Logging Global Process records
    Device= Disk FileSys records
Thresholds: CPU= 10.00%, Disk=10.0/sec, First=5.0 sec, Resp=30.0 sec,
    Trans=100 Nonew=FALSE, Nokilled=FALSE, Shortlived=FALSE
    (<1 sec)
HP-UX Parms: Buffer Cache Size = 16384KB, NPROC = 532
Wait Thresholds: CPU=100.00%, Memory=100.00%
Impede=100.00%
Memory: Physical = 84.0 MB, Swap = 124304.0 MB, Available to users = 66.5 MB. There
are 2 LAN interfaces: 0, 1.
06/03/99 15:28 There are 2 disk devices:
    Disk #1976          = "/dev/hdisk0"
    Disk #1987          = "/dev/hdisk1"
```

The date and time listed on the first line correspond to the *first date and time* in the global log file and indicate when *scope* was started. Data records may have been rolled out of the global log file so the date and time on this report do not necessarily indicate the *first global record* in the log file.

Initial Parm File Application Definitions

To obtain this report, use the `scan` command with its default `detail on` and have application data in the log file.

This report lists the name and definition of each application at the time the first application record is listed in the log file. Any application addition or deletion notifications you receive are based on this initial list of applications. For example:

```
06/01/99 08:39 Application(1) = "other"
Comment=all processes not in user-defined applications
06/01/99 08:39 Application(2) = "Real_TimeSystem"
Priority range = 0-127
06/01/99 08:39 Application(3) = "Prog_Development"
File=vi,ed,sed,xdb,ld,lint,cc,ccom,pc,pascomp
```

During the scan, you are notified of applications that were added or deleted. Additions and deletions are determined by comparing the spelling and case of the old application names to the new set of logged application names. No attempt is made to detect a change in the definition of an application. If an application with a new name is detected, it is listed along with its new definition. The date and time on this record is the last time *scope* was started before logging the first application record currently in the log file.

Chronological Detail

This section describes the following chronological details:

- `parm` file global change notifications
- `parm` file application addition and deletion notifications
- `scope` off-time notifications
- Application-specific summary report

Parm File Global Change Notifications

To obtain this report, use the `scan` command with its default `detail on`.

This report is generated any time a record is found that *scope* started.

The following example shows the change notifications that occur when two new disk drives are added to the system.

```
03/13/99 17:30 The number of disk drives changed from 9 to 11
03/13/99 17:30 New disk device scsi-4 = "c4d0s*"
03/13/99 17:30 New disk device scsi-3 = "c3d0s*"
```

Parm File Application Addition/Deletion Notifications

To obtain this report, use the `scan` command with its default `detail on` and have application data in the log file.

User-defined applications can be added or deleted each time `scope` is started. If an application name is found that does not match the last set of applications, an application addition, deletion, or change notification is printed. If the name of an application has not changed, it is not printed.

The following example shows that a new application was started.

```
03/13/99 17:30 Application 4 "Accounting_Users_1" was added
User=ted,rebecca,test*,mark,gene
```

Application definitions are not checked for changes. They are listed when an application name is changed, but any change to an existing application's definition without an accompanying name change is not detected.

scope Off-Time Notifications

To obtain this report, use the `scan` command with its default `detail on`.

If an extracted file contains only summary information, times are rounded to the nearest hour. For example:

```
06/03/99 11:00 - 06/03/99 12:34 collector off (01:34:04)
```

The first date and time (06/03/99 11:00) indicates the last valid data record in the log file before `scope` was restarted. The second date and time (06/03/99 12:34) indicates when `scope` was restarted.

The last field (in parentheses) shows how long `scope` was *not* running. The format is `ddd/hh:mm:ss`, where `ddd` are days and `hh:mm:ss` are hours, minutes, and seconds. Zeros to the left are deleted.

In this example, `scope` was off on June 3, 1999 between 11:00 am and 12:34 pm. The summary information shows that data was not collected for one hour, 34 minutes, and four seconds.

Application-Specific Summary Report

To obtain this report, use the `scan` command with its default `detail on` and have application data in the log file.

This report can help you define applications. Use the report to identify applications that are accumulating either too many or too few system resources and those that could be consolidated with other applications. Applications that accumulate too many system resources might benefit by being split into multiple applications.

You should define applications in ways that help you make decisions about system performance tuning. It is unlikely that system resources will accumulate evenly across applications.

The application-specific summary report is generated whenever the application definitions change to allow you to access the data of the application definitions before and after the change.

A final report is generated for all applications. This report covers only the time since the last report and not the entire time covered by the log file. For example:

Application	PERCENT OF TOTAL			
	Records	CPU	DISK	TRANS
-----	-----	-----	-----	-----
OTHER	22385	45.7%	20.9%	63.0%
Resource_Sharing	7531	6.0%	2.2%	17.1%
SPOOLING	13813	2.4%	0.3%	0.0%
ON_LINE_COMPILE	13119	2.9%	1.7%	0.1%
BATCH_COMPILE	8429	2.9%	0.1%	2.2%
ORDER_ENTRY	387	0.1%	0.0%	0.0%
ELECTRONIC_MAIL	6251	3.8%	1.3%	9.6%
PROGRAM_DEVELOPMENT	3141	9.1%	2.4%	0.6%
RESEARCH_DEPARTMENT	3968	8.7%	2.0%	6.0%
BILL_OF_MATERIALS	336	0.6%	1.5%	0.1%
FINANCIALS	1080	5.0%	1.5%	0.5%
MARKETING_DEPT	2712	12.9%	67.3%	0.0%
GAMES	103	0.1%	0.0%	0.0%
-----	-----	-----	-----	-----
All user applications	73.1%	54.3%	79.1%	37.0%

Summaries

This section describes the following summaries:

- Process log reason summary
- Scan start and stop actual dates and times
- Application overall summary
- scope coverage summary
- Log file contents summary
- Log file empty space summary

Process Log Reason Summary

To obtain this report, you must have process data in the log file.

This report helps you set the interesting process thresholds for scope. The report lists every reason a process might be considered interesting and thus get logged, along with the total number of processes logged that satisfied each condition.

The following example shows a process log reason summary report:

Process Summary Report: 04/13/99 3:32 PM to 05/04/99 6:36 PM

There were 93.8 hours of process data

Process records were logged for the following reasons:

Log Reason	Records	Percent	Recs/hr
-----	-----	-----	-----
New Processes	17619	53.9%	44.7
Killed Processes	16047	49.1%	40.7
CPU Threshold	3169	9.7%	8.0
Disk Threshold	1093	3.3%	2.8

NOTE: A process can be logged for more than one reason at a time. Record counts and percentages will not add up to 100% of the process records.

If the `detail on` command is issued, this report is generated each time a threshold value is changed so you can evaluate the effects of that change. Each report covers the period since the last report. A final report, generated when the scan is finished, covers the time since the last report.

If the `detail off` command is issued, only one report is generated covering the entire scanned period.

You can reduce the amount of process data logged by `scope` by modifying the `parm` file's `threshold` parameter and raising the thresholds of the interest reasons that generate the most process log records. To increase the amount of data logged, lower the threshold for the area of interest.

In the previous example, you could decrease the amount of disk space used for the process data (at the expense of having less information logged) by raising the `CPU` threshold or setting the `nonew` threshold.

Scan Start and Stop

This summary report is printed if any valid data was scanned. It gives actual dates and times that the scan was started and stopped. For example:

```
Scan started on      03/03/99 12:40 PM
Scan stopped on     03/11/99  1:25 PM
```

Application Overall Summary

To obtain this report, you must have application data in the log file.

This report is an overall indicator of how much system activity is accumulated in user-defined applications, rather than in the other application. If a significant amount of a critical resource is not being captured by user applications, you might consider scanning the process data for processes that can be included in user applications.

For example:

```
OVERALL, USER DEFINED APPLICATIONS ACCOUNT FOR
      82534 OUT OF   112355 RECORDS   ( 73.5%)
      218.2 OUT OF    619.4 CPU HOURS ( 35.2%)
       24.4 OUT OF    31.8 M DISC IOS ( 76.8%)
        0.2 OUT OF     0.6 M TRANS   ( 27.3%)
```

Collector Coverage Summary

This report is printed if any valid global or application data was scanned. It indicates how well scope is being used to collect system activity. If the percentage of time scope was off is high, as in the example below, you should review your operational procedures for starting and stopping scope.

The total time covered was 108/16:14:51 out of 128/00:45:02

Time lost when collector was off 19/08:30:11 15.12%

The scopeux collector was started 45 times

This report will be more complete if global detail data is included in the scan. If only summary data is available, you determine the time scope was stopped and started only to the nearest hour. (An appropriate warning message is printed with the report if this is the case.)

The total time covered is determined by accumulating all the interval times from the logged data. The out of time value is calculated by subtracting the starting date and time from the ending date and time. This should represent the total time that could have been logged. The "Time lost when collector was off" value is the total time less the covered time.

The formats for the three times mentioned are:

ddd/hh:mm:ss

where *ddd* are days and *hh:mm:ss* are hours, minutes, and seconds.

In the previous example, the total time collected was 108 days, 16 hours, 14 minutes, and 51 seconds.

Log File Contents Summary

The log file contents summary is printed *if any* valid data was scanned. It includes the log file space and the dates covered. This summary is helpful when you are resizing your log files with the `resize` command.

-----Total-----		-----Each Full Day-----		-----Dates-----		Full	
Type	Records	MBytes	Records	MBytes	Start	Finish	Days
Global	1376	0.27	288.9	0.057	05/23/99 to	05/28/99	4.8
Application	6931	0.72	1455.0	0.152	05/23/99 to	05/28/99	4.8
Process	7318	1.14	1533.6	0.239	05/23/99 to	05/28/99	4.8
Disk	2748	0.07	567.6	0.014	05/23/99 to	05/28/99	4.8
Transaction	no data found						
Overhead		0.29					
	-----	-----	-----	-----			
TOTAL	18373	2.49	3845.0	0.461			

The columns are described as follows:

Column	Explanation
Type	The general type of data being logged. One special type, <i>Overhead</i> , exists: <i>Overhead</i> is the amount of disk space occupied (or reserved) by the log file <i>versus</i> the amount actually used by the scanned data records. If less than the entire log file was scanned, <i>Overhead</i> includes the data records that were not scanned. If the entire file was scanned, <i>Overhead</i> accounts for any inefficiencies in blocking the data into the file <i>plus</i> any file-access support structures. It is normal for extracted log files to have a higher overhead than raw log files since they have additional support structures for quicker positioning.
Total	The total record count and disk space scanned for each type of data.
Each Full Day	The number of records and amount of disk space used for each 24-hour period that <i>scope</i> runs.
Dates	The first and last valid dates for the data records of each data type scanned.
Full Day	The number of full (24-hour) days of data scanned for this data type. <i>Full Days</i> may not be equal to the difference between the start and stop dates if <i>scope</i> coverage did not equal 100 percent of the scanned time.

The TOTAL line (at the bottom of the listed data) gives you an idea of how much disk space is being used and how much data you can expect to accumulate each day.

Log File Empty Space Summary

This summary is printed for each log file scanned. For example:

```
The Global      file is now 13.9% full with room for 61 more full days
The Application file is now 15.1% full with room for 56 more full days
The Process     file is now 23.5% full with room for 32 more full days
The Device      file is now  1.4% full with room for 2896 more full days
```

The amount of room available for more data is calculated based on the following factors:

- The amount of unused space in the file
- The scanned value for the number of megabytes of data being logged on each 24-hour day

If the megabytes-scanned-per-day values appear unrealistically low, they are replaced with default values for this calculation.

If you scan an extracted file, you get a single report line because all data types share the same extracted file.

Chapter 6: Utility Commands

This chapter describes the **utility** program's commands. It includes a syntax summary and a command reference section that lists the commands in alphabetical order.

Utility commands and parameters can be entered with any combination of uppercase and lowercase letters. Only the first three letters of the command name are required. For example, the logfile command can be entered as **logfile** or it can be abbreviated as **log** or **LOG**.

Examples of how these commands are used can be found in online help for the **utility** program.

The table on the next pages contains a summary of **utility** command syntax and parameters.

Table 4 Utility Commands: Syntax and Parameters

Command	Parameter
analyze	
checkdef	alarmdef file
detail	on off
exit e	
guide	
list	<i>filename</i> or *
logfile	logfile
menu ?	
parmfile	parmfile
quit q	
resize	global application process device transaction days=maxdays

Command	Parameter
	size=max MB empty=days space=MB yes no maybe
scan	<i>logfile</i> (Operation is also affected by the list, start, stop, and detail commands.)
show	all
sh !	system command
start	<i>date [time]</i> today [-days] [time] last [-days] [time] first [+days] [time]
stop	<i>date [time]</i> today [-days] [time] last [-days] [time] first [+days] [time]

analyze

Use the analyze command to analyze the data in a log file against alarm definitions in an alarm definitions (**alarmdef**) file and report resulting alarm status and activity. Before issuing the analyze command, you should run the checkdef command to check the alarm definitions syntax. Checkdef also sets and saves the alarm definitions file name to be used with analyze. If you do not run checkdef before analyze, you are prompted for an alarm definitions file name.

If you are using command line mode, the default alarm definitions file **/var/opt/perf/alarmdef** is used.

For detailed information about alarm definitions, see [Performance Alarms](#).

Syntax

analyze

How to Use It

When you issue the analyze command, it analyzes the log files specified in the data sources configuration file, **datasources**, against the alarm definitions in the **alarmdef** file.

The **analyze** command allows you to evaluate whether or not your alarm definitions are a good match against the historical data collected on your system. It also lets you decide if your alarm definitions will generate too many or too few alarms on your analysis workstation.

Also, you can perform data analysis with definitions (IF statements) set in the alarm definitions file because you can get information output by PRINT statements when conditions are met. For explanations of how to use the IF and PRINT statements in an alarm definition, see [Performance Alarms](#).

You can optionally run the **start**, **stop**, and **detail** commands with **analyze** to customize the analyze process. You specify these commands in the following order:

```
checkdef
start
stop
detail
analyze
```

Use the start and stop commands if you want to analyze log file data that was collected during a specific period of time. (Descriptions of the start and stop commands appear later in this chapter.)

While the analyze command is executing, it lists alarm events such as alarm start, end, and repeat status plus any text in associated print statements. Also, any text in PRINT statements is listed as conditions (in IF statements) become true. EXEC statements are not executed but are listed so you can see what would have been executed. An alarm summary report shows a count of the number of alarms and the amount of time each alarm was active (on). The count includes alarm starts and repeats, but not alarm ends.

If you want to see the alarm summary report only, issue the detail off command. However, if you are using command line mode, detail off is the default so you need to specify -D to see the alarm events as well as the alarm summary.

Example

The checkdef command checks the alarm definitions syntax in the **alarmdef** file and saves the name of the **alarmdef** file for later use with the analyze command. The start today command specifies that only data logged today is to be analyzed. Lastly, the analyze command analyzes the log file in the default **SCOPE** data source specified in the **datasources** file against the alarm definitions in the **alarmdef** file.

```
utility>
checkdef /var/opt/perf/alarmdef
start today
analyze
```

To perform the above task using command line arguments, enter:

```
utility -xc -D -b today -xa
```

checkdef

Use the checkdef command to check the syntax of the alarm definitions in an alarm definitions file and report any warnings or errors that are found. This command also sets and saves the alarm definitions

file name for use with the analyze command.

For descriptions of the alarm definitions syntax and how to specify alarm definitions, see [Performance Alarms](#).

Syntax

```
checkdef [/directorypath/alarmdef]
```

Parameters

alarmdef	The name of any alarm definitions file. This can be a user-specified file or the default alarmdef file. If no directory path is specified, the current directory will be searched.
----------	---

How to Use It

When you have determined that the alarm definitions are correct, you can process them against the data in a log file using the analyze command.

In batch mode, if no alarm definitions file is specified, the default alarmdef file is used.

In interactive mode, if no alarm definitions file is specified, you are prompted to specify one.

Example

The checkdef command checks the alarm definitions syntax in the **alarmdef** file and then saves the name of the **alarmdef** file for later use with the analyze command.

```
utility>  
checkdef /var/opt/perf/alarmdef
```

To perform the above task using command line arguments, enter:

```
utility -xc
```

detail

Use the detail command to control the level of detail printed in the analyze, parmfile, and scan reports.

The default is detail on in interactive and batch modes and detail off in command line mode.

Syntax

detail	[on]
	[off]

Parameters

on	Prints the effective contents of the parm file as well as parm file errors. Prints complete analyze and scan reports.
off	In the parm file report, application definitions are <i>not</i> printed. In the scan report, scope

collection times, initial parm file global information, and application definitions are <i>not</i> printed. In the analyze report, alarm events and alarm actions are <i>not</i> printed.
--

How to Use It

For explanations of how to use the detail command with the analyze, scan, and parmfile commands, see the [analyze](#), [parmfile](#) and [scan](#) command descriptions in this chapter.

Examples

For examples of using the detail command, see the descriptions of the [analyze](#), [parmfile](#), and [scan](#) commands in this chapter.

exit

Use the exit command to terminate the **utility** program. The exit command is equivalent to the **utility** program's quit command.

Syntax

exit

e

guide

Use the guide command to enter guided commands mode. The guided command interface leads you through the various **utility** commands and prompts you to perform the most common tasks that are available.

Syntax

guide

How to Use It

- To enter guided commands mode from **utility**'s interactive mode, type **guide** and press **Return**.
- To accept the default value for a parameter, press **Return**.
- To terminate guided commands mode and return to interactive mode, type **q** at the guide> prompt.

This command does not provide all possible combinations of parameter settings. It selects settings that should produce useful results for the majority of users.

help

Use the help command to access the **utility** program's online help facility.

Syntax

help [*keyword*]

How to Use It

You can enter parameters to obtain information on **utility** commands and tasks, or on help itself. You can navigate to different topics by entering a key word. If more than one page of information is available, the display pauses and waits for you to press **Return** before continuing. Type **q** or **quit** to exit the help system and return to the **utility** program.

You can also request help on a specific topic. For example,

```
help tasks
```

or

```
help resize parmfile
```

When you use this form of the help command, you receive the help text for the specified topic and remain in the **utility** command entry context. Because you do not enter the help subsystem interactively, you do not have to type **quit** before entering the next **utility** command.

list

Use the list command to specify the output file for all **utility** reports. The contents of the report depend on which other commands are issued after the list command. For example, using the list command before the logfile, detail on, and scan commands produces the list file for a detailed summary report of a log file.

Syntax

```
list [filename] |*
```

where * sets the output back to stdout.

How to Use It

There are two ways to specify the list file for reports:

- Redirect stdout when invoking the **utility** program by typing:
utility > utilrept
- Or, use the list command when **utility** is running by typing:
list utilrept

In either case, user interactions and errors are printed to stderr, and reports go to the file specified.

The *filename* parameter in the list command must represent a valid filename to which you have write access. Existing files have the new output appended to the end of existing contents. If the file does not exist, it will be created.

To determine the current output file, issue the list command without parameters:

If the output file is not stdout, most commands are echoed to the output file as they are entered.

Example

The list command produces a summary report on the extracted log file rxlog. The list utilrept command directs the scan report listing to a disk file. Detail off specifies less than full detail in the report. The scan command reads rxlog and produces the report.

The list * command sets the list device back to the default stdout. !lp utilrept sends the disk file to the system printer.

```
utility>  
logfile rxlog  
list utilrept  
detail off  
scan  
list *  
!lp utilrept
```

To perform the above task using command line arguments, enter:

```
utility -l rxlog -f utilrept -d -xs print utilrept
```

logfile

Use the logfile command to open a log file. For many **utility** program functions, a log file must be opened. You do this explicitly by issuing the logfile command or implicitly by issuing some other command. If you are in batch or command line mode and do not specify a log file name, the **default /var/opt/perf/datafiles/logglob** file is used. If you are in interactive mode and do not specify a log file name, you are prompted to provide one or accept the default **/var/opt/perf/datafiles/logglob** file.

Syntax

logfile [*logfile*]

How to Use It

You can specify the name of either a raw or extracted log file. If you specify an extracted log file name, all information is obtained from this single file. You do not need to specify any of the raw log files other than the global log file, logglob. Opening **logglob** gives you access to all of the data in the other logfiles.

Raw log files have the following names:

logglob	global log file
logappl	application log file
logproc	process log file
logdev	device log file
logtran	transaction log file

logls	logical systems log file
logindx	index log file

Once a log file is opened successfully, a report is printed or displayed showing the general content of the log file (or log files), as shown in the example below.

```
Global file: /var/opt/perf/datafiles/logglob version DApplication file:
/var/opt/perf/datafiles/logapplProcess file: /var/opt/perf/datafiles/logprocDevice file:
/var/opt/perf/datafiles/logdevTransaction file: /var/opt/perf/datafiles/logtranIndex file:
/var/opt/perf/datafiles/logindxSystem ID: homerSystem Type 9000/715 S/N 6667778899 O/S HP-UX
B.10.20. AData Collector: SCOPE/UX C.02.30File Created: 06/14/99Data Covers: 27 days to
7/10/99Shift is: All DayData records available are: Global Application Process Disk Volume
TransactionMaximum file sizes: Global=10.0 Application=10.0 Process=20.0 Device=10.0
Transaction=10.0 MB
```

```
The first GLOBAL record is on 06/14/99 at 12:00 AMThe first APPLICATION
record is on 06/25/99 at 12:00 AMThe first PROCESS record is on 07/06/99 at
12:01 AMThe first DEVICE record is on 05/01/99 at 11:50 AMThe first
TRANSACTION record is on 05/01/99 at 11:55 AM The default starting date &
time = 05/01/99 11:50 AM (FIRST + 0)The default stopping date & time =
07/10/99 11:59 PM (LAST - 0)
```

You can verify the log file you opened with the show command, as described later.

You can open another log file at any time by entering another logfile command. Any currently opened log file is closed before the new log file is opened.

The resize and scan commands require a log file to be open. If no log file is currently open, an implicit logfile command is executed.

Note: Do not rename raw log files. Access to these files assumes that the standard log file names are in effect.

Note: If you must have more than one set of raw log files on the same system, create a separate directory for each set of files. Although the log file names cannot be changed, different directories may be used. If you want to resize the log files in any way, you must have read/write access to all the log files.

menu

Use the menu command to print a list of the available **utility** commands.

Syntax

```
menu
```

Example

```
utility> menu
```

Command	Parameters	Function
HELP	topic]	Get information on commands and options
GUIDE		Enter guided commands mode for novice users
LOGFILE	[logname]	Specify a log file to be processed
LIST	[filename *]	Specify the listing file
START	[startdate time]	Set starting date & time for SCAN or ANALYZE
STOP	[stopdate time]	Set ending date & time for SCAN or ANALYZE
DETAIL	[ON OFF]	Set report detail for SCAN, PARMFILE, or ANALYZE
SHOW	[ALL]	Show the current program settings
PARMFILE	[parmfile]	Check parsing of a parameter file
SCAN	[logname]	Read the log file and produce a summary report
RESIZE	[GLOB APPL PROC DEV TRAN] [DAYS=][EMPTY=]	Resize raw log files
CHECKDEF	[alarmdef]	Check parsing and set the alarmdef file
ANALYZE		Analyze the log file using the alarmdef file
! or Sh	[command]	Execute a system command
MENU or ?		List the commands menu (This listing)
EXIT or Q		Terminate the program
utility>		

parmfile

Use the parmfile command to view and syntax check the Performance Collection Component parm file settings that are used for data collection.

Syntax

parmfile [/directorypath/parmfile]

How to Use It

You can use the parmfile command to do any of the following:

- Examine the **parm** file for syntax warnings and review the resulting settings. All parameters are checked for correct syntax and errors are reported. After the syntax check is completed, only the applicable settings are reported.
- Find out how much room is left for defining applications.
- If detail on is specified, print the effective contents of the **parm** file plus any default settings that were not overridden, and print application definitions.

In batch mode, if no **parm** file name is specified, the default **parm** file is used.

In interactive mode, if no **parm** file name is supplied, you are prompted to supply one.

Example

The **parmfile** command checks the syntax of the current **parm** file and reports any warnings or errors.

Detail on lists the logging parameter settings.

```
utility>  
detail on  
parmfile parm
```

To perform the above task using command line arguments, enter:

```
utility -xp -D
```

quit

Use the **quit** command to terminate the **utility** program. The **quit** command is equivalent to the **utility** program's exit command.

Syntax

```
quit
```

```
q
```

resize

Use the **resize** command to manage the space in your raw log file set. This is the *only* program you should use to resize the raw log files in order to preserve coordination between the files and their internal control structures. If you use other tools you might remove or destroy the validity of these control structures.

The **utility** program *cannot* be used to resize extracted files. If you want to resize an extracted file, use the **extract** program to create a new extracted log file.

Syntax

resize	[global] [application] [process] [device] [transaction]	[days=maxdays] [size=maxMB]	[empty=days] [space=MB]	[yes] [no] [maybe]
--------	--	---------------------------------	-----------------------------	----------------------------

Parameters

log file type	Specifies the type of raw data you want to resize: global, application, process, device, or transaction, which correspond to the raw log files logglob , logappl , logproc , logdev , and logtran . If you do not specify a data type and
---------------	--

	are running utility in batch mode, the batch job terminates. If you are running utility interactively, you are prompted to supply the data type based on those log files that currently exist.
days & size	Specify the maximum size of the log file. The actual size depends on the amount of data in the file.
empty & space	Specify the minimum amount of room required in the file after the resizing operation is complete. This value is used to determine if any of the data currently in the log file must be removed in the resizing process.

You might expect that a log file would not fill up until the specified number of days after a resizing operation. You may want to use this feature of the resize command to minimize the number of times a log file must be resized by the scope collector because resizing can occur any time the file is filled. Using resize to force a certain amount of empty space in a log file causes the log file to be resized when you want it to be.

The days and empty values are entered in units of days; the size and space values are entered in units of megabytes. Days are converted to megabytes by using an average megabytes-per-day value for the log file. This conversion factor varies depending on the type of data being logged and the particular characteristics of your system.

More accurate average-megabytes-per-day conversion factors can be obtained if you issue the scan command on the existing log file before you issue the resize command. A scan measures the accumulation rates for your system. If no scan is done or if the measured conversion factor seems unreasonable, the resize command uses a default conversion factor for each type of data.

yes	Specifies that resizing should be unconditionally performed. This is the default action if utility is not running interactively. If no action is specified when utility is running interactively, you are prompted to supply the action.
no	Specifies that resizing should not be performed. This parameter can be specified as an action if you want to see the resizing report but do not want to perform the resizing at that time.
maybe	Specifies that utility should decide whether or not to resize the file. This parameter forces utility to make this decision based on the current amount of empty space in the log file (before any resizing) and the amount of space specified in the resize command. If the current log file contains at least as much empty space as specified, resizing does <i>not</i> occur. If the current log file contains less than the specified empty space, resizing occurs.
maybe (continued)	If the resizing can be made without removing any data from the log file (for example, increasing the maximum log file size, or reducing the maximum log file size without having to remove any existing data), resizing occurs. The maybe parameter is intended primarily for use by periodic batch executions. See the “Examples” subsection below for an explanation of how to use the resize command in this manner.

Default resizing parameters are shown in the following table.

Table 5 Default Resizing Parameters

Parameter	If Executed Interactively	If Executed in Batch
log file type	You are prompted for each available log file type.	No default. This is a required parameter.
dayssize	The current file size.	The current file size.
empty space	The current amount of empty space or enough empty space to retain all data currently in the file, whichever is smaller.	The current amount of empty space or enough empty space to retain all data currently in the file, whichever is smaller.
yesnomaybe	You are prompted following the reported disk space results.	Yes. Resizing will occur.

How to Use It

Before you resize a log file, you *must* stop Performance Collection Component using the steps under [Stopping and Restarting Data Collection](#) in Chapter 2.

A raw log file must be opened before resizing can be performed. Open the raw log file with the logfile command before issuing the resize command. The files cannot be opened by any other process.

The resize command creates the new file scopelog in the directory set by TMPDIR environment variable before deleting the original log file. If the environment variable TMPDIR is not set, then the /var/tmp directory (/tmp on IBM AIX 4.1 and later) will be used as temporary location. Make sure there is sufficient disk space in the directory specified by the TMPDIR or in the /var/tmp directory (/tmp on IBM AIX 4.1 and later) to hold the original log file before doing the resizing procedure.

After resizing, a log file consists of data plus empty space. The data retained is calculated as the maximum file size minus the required empty space. Any data removed during the resizing operation is lost. To save log file data for longer periods, use extract to copy this data to an extracted file *before* doing the resize operation.

Resize Command Reports

One standard report is produced when you resize a raw log file. It shows the three interrelated disk space categories of maximum file size, data records, and empty space, before and after resizing. For example:

```
resize global days=120;empty=10
empty space raised to match file size and data records
final resizing parameters:
file: logglob                megabytes / day: 0.101199
      ---currently-----   --after resizing---
maximum size:  65 days (  6.6 mb)  120 days ( 12.1 mb)  83% increase
data records:  61 days (  6.2 mb)   61 days (  6.2 mb)  no data removed
```

```
empty space:      4 days ( 0.5 mb)   59 days ( 6.0 mb)  1225% increase
```

The megabytes per day value is used to convert between days and megabytes. It is either the value obtained during the scan function or a default for the type of data being resized.

The far right-hand column is a summary of the net change in each category of log file space. Maximum size and empty space can increase, decrease, or remain unchanged. Data records have either no data removed or a specified amount of data removed during resizing.

If the resize is done interactively and one or more parameters are defaults, you can get a preliminary resizing report. This report summarizes the current log file contents and any parameters that were provided. The report is provided to aid in answering questions on the unspecified parameters. For example:

```
resize global days=20

file resizing parameters (based on average daily
space estimates and user resizing parameters)

file: logglob                                megabytes / day: 0.101199

      -----currently-----      --after resizing--
maximum size:  65 days ( 6.6 mb)   20 days ( 2.0 mb)
data records:  61 days ( 6.2 mb)   ??
empty space:   4 days ( 0.5 mb)    ??
```

In this example, you are prompted to supply the amount of empty space for the file before the final resizing report is given. If no action parameter is given for interactive resizing, you are prompted for whether or not to resize the log file immediately following the final resizing report.

Examples

The following commands are used to resize a raw process log file. The scan is performed before the resize to increase the accuracy of the number-of-days calculations.

```
logfile /var/opt/perf/datafiles/logglob
detail off
scan
resize process days=60 empty=30 yes
```

days=60 specifies holding a maximum of 60 days of data. empty=30 specifies that 30 days of this file are currently empty. That is, the file is resized with no more than 30 days of data in the file to leave room for 30 more days out of a total of 60 days of space. yes specifies that the resizing operation should take place regardless of current empty space.

The next example shows how you might use the resize command in batch mode to ensure that log files do not fill up during the upcoming week (forcing scope to resize them). You could schedule a cron script using the at command that specifies a minimum amount of space such as 7 days - or perhaps 10 days, just to be safe.

The following shell script accomplishes this:

```
echo detail off                                > utilin
```

```
echo scan >> utilin
echo resize global empty=10 maybe >> utilin
echo resize application empty=10 maybe >> utilin
echo resize process empty=10 maybe >> utilin
echo resize device empty=10 maybe >> utilin
echo quit >> utilin
utility < utilin > utilout 2> utilerr
```

Specifying `maybe` instead of `yes` avoids any resizing operations if 10 or more days of empty space currently exist in any log files. Note that the maximum file size defaults to the current maximum file size for each file. This allows the files to be resized to new maximum sizes without affecting this script.

scan

Use the `scan` command to read a log file and write a report on its contents. (For a detailed description of the report, see [Utility Scan Report Details](#).)

Syntax

`scan`

How to Use It

The `scan` command requires a log file to be opened. The log file scanned is the first of one of the following:

- The log file named in the `scan` command itself.
- The last log file opened by any previous command.
- The default log file.

In this case, interactive users are prompted to override the default log file name if desired.

The following commands affect the operation of the `scan` function:

<code>detail</code>	Specifies the amount of detail in the report. The default, <code>detail on</code> , specifies full detail.
<code>list</code>	Redirects the output to another file. The default is to list to the standard list device.
<code>start</code>	Specifies the date and time of the first log file record you want to scan. The default is the beginning of the log file.
<code>stop</code>	Specifies the date and time of the last log file record you want to scan. The default is the end of the log file.

For more information about the `detail`, `list`, `start`, and `stop` commands, see their descriptions in this chapter.

The scan command report consists of 12 sections. You can control which sections are included in the report by issuing the detail command prior to issuing scan.

The following four sections are always printed (even if detail off is specified):

- Scan start and stop actual dates and times
- Collector coverage summary
- Log file contents summary
- Log file empty space summary

The following sections are printed if detail on (the default) is specified:

- Initial **parm** file global information and system configuration information
- Initial **parm** file application definitions
- **parm** file global changes
- parm file application addition/deletion notifications
- Collector off-time notifications
- Application-specific summary reports

The following section is always printed if application data was scanned (even if detail off is specified):

- Application overall summary

The following section is always printed if process data was scanned (even if detail off is specified):

- Process log reason summary

Example

The scan of the current default global log file starts with records logged from June 1, 1999 at 7:00 AM until the present date and time.

```
utility>  
logfile /var/opt/perf/datafiles/logglob  
detail on  
start 6/1/99 7:00 am  
scan
```

To perform the above task using command line arguments, enter:

```
utility -D -b 6/1/99 7:00 am -xs
```

sh

Use sh to enter a shell command without exiting **utility** by typing **sh** or an exclamation point (!) followed by a shell command.

Syntax

sh or ! **[shell command]**

Parameters

sh ls	Executes the ls command and returns to utility .
-------	---

!ls	Same as above.
-----	----------------

How to Use It

Following the execution of the single command, you automatically return to **utility**. If you want to issue multiple shell commands without returning to **utility** after each one, you can start a new shell. For example,

```
sh ksh
```

or

```
!ksh
```

show

Use the show command to list the names of the files that are open and the status of the **utility** parameters that can be set.

Syntax

```
show [all]
```

Examples

Use show to produce a list that may look like this:

```
Logfile: /var/opt/perf/datafiles/logglob List: "stdout"Detail: ON for  
ANALYZE, PARMFILE and SCAN functions
```

```
The default starting date & time = 10/08/99 08:17 AM (FIRST + 0)The default  
stopping date & time = 11/20/99 11:59 PM (LAST - 0)The default shift = 12:00 AM -  
12:00 AM
```

Note: The default shift time is shown for information only. Shift time cannot be changed in utility.

Use show all to produce a more detailed list that may look like this:

```
Logfile: /var/opt/perf/datafiles/logglob  
  
Global      file: /var/opt/perf/datafiles/logglob  
Application file: /var/opt/perf/datafiles/logappl  
Process     file: /var/opt/perf/datafiles/logproc  
Device      file: /var/opt/perf/datafiles/logdev  
Transaction file: /var/opt/perf/datafiles/logtran  
Index       file: /var/opt/perf/datafiles/logindx  
System ID:  homer  
System Type 9000/715 S/N 66677789 OS/ HP-UX B.10.20 A  
Data Collector: SCOPE/UX C.02.30  
File created: 10/08/99  
Data Covers: 44 days to 11/20/99
```

```

Shift is:      All Day
Data records available are:
  Global Application Process Disk Volume Transaction

Maximum file sizes:
  Global=10.0 Application=10.0 Process=20.0 Device=10.0 Transaction 10.0 MB

List      "stdout"
Detail   ON for ANALYZE, PARMFILE and SCAN functions

The default starting date & time = 10/08/99 11:50 AM (FIRST + 0)
The default stopping date & time = 11/20/99 11:59 PM (LAST - 0)
The default shift = 12:00 AM - 12:00 AM

```

start

Use the start command to specify the beginning of the subset of a log file that you want to scan or analyze. Start lets you start the scan or analyze process at data that was logged at a specific date and time.

The default starting date and time is set to the date and time of the first record of any type in a log file that has been currently opened with the logfile command.

Syntax

start	<i>[date</i>	<i>[time]]</i>	
	[today	<i>[-days]</i>	<i>[time]]</i>
	[last	<i>[-days]</i>	<i>[time]]</i>
	[first	<i>][+days]</i>	<i>[time]]</i>

Parameters

date	The date format depends on the native language configured on the system being used. If you do not use native languages or have the default language set to C, the date format is <i>mm/dd/yy</i> (month/day/year) or 06/30/99 for June 30, 1999.
time	The time format also depends on the native language being used. For C, the format is <i>hh:mm am</i> or <i>hh:mm pm</i> (hour:minute in 12-hour format with the am/pm suffix) such as 07:00 am for 7 o'clock in the morning. Twenty-four hour time is accepted in all languages. For example, 23:30 would be accepted for 11:30 pm. If the date or time is entered in an unacceptable format, an example in the correct format is shown. If no start time is given, a midnight (12 am) is assumed. A starting time of midnight for a given day starts at the <i>beginning</i> of that day (00:00 on a 24-hour clock).
today	Specifies the current day. The parameter <i>today-days</i> specifies the number of days prior to today's date. For example, <i>today-1</i> indicates yesterday's date and <i>today-2</i> indicates the day before yesterday.

<code>last</code>	Can be used to represent the last date contained in the log file. The parameter <code>last-days</code> specifies the number of days prior to the last date in the log file.
<code>first</code>	Can be used to represent the first date contained in the log file. The parameter <code>first+days</code> specifies the number of days after the first date in the log file.

How to Use It

The start command is useful if you have a very large log file and do not want to scan or analyze the entire file. You can also use it to specify a specific time period for which data is scanned. For example, you can scan a log file for data that was logged for a period beginning 14 days before the present date by specifying **today-14**.

You can use the stop command to further limit the log file records you want to scan.

If you are not sure whether native language support is installed on your system, you can force **utility** to use the C date and time formats by issuing the following statement before running **utility**:

```
LANG=C; export LANG
```

or in C Shell

```
setenv LANG C
```

Example

The scan of the default global log file starts with records logged from August 5, 1999 at 8:00 AM until the present date and time.

```
utility>
logfile /var/opt/perf/datafiles/logglob
detail on
start 8/5/99 8:00 AM
scan
```

To perform the above task using command line arguments, enter:

```
utility -D -b 8/5/99 8:00 am -xs
```

stop

Use the stop command to specify the end of a subset of a log file that you want to scan or analyze. Stop lets you terminate the scan or analyze process at data that was logged at a specific date and time.

The default stopping date and time is set to the date and time of the last record of any type in a log file that has been currently opened with the logfile command.

Syntax

stop	<i>[date</i>	<i>[time]]</i>	
	[today	<i>[-days]</i>	<i>[time]]</i>
	[last	<i>[-days]</i>	<i>[time]]</i>
	[first	<i>[+days]</i>	<i>[time]]</i>

Parameters

date	The date format depends on the native language configured on the system being used. If you do not use native languages or have the default language set to C, the date format is <i>mm/dd/yy</i> (month/day/year) or 06/30/99 for June 30, 1999.
time	The time format also depends on the native language being used. For C, the format is <i>hh:mm am</i> or <i>hh:mm pm</i> (hour:minute in 12-hour format with the am/pm suffix) such as 07:00 am for 7 o'clock in the morning. Twenty-four hour time is accepted in all languages. For example, 23:30 would be accepted for 11:30 pm. If the date or time is entered in an unacceptable format, an example in the correct format is shown. If no stop time is given, one minute before midnight (11:59 pm) is assumed. A stopping time of midnight (12 am) for a given day stops at the <i>end</i> of that day (23:59 on a 24-hour clock).
today	Specifies the current day. The parameter <i>today-days</i> specifies the number of days prior to today's date. For example, <i>today-1</i> indicates yesterday's date and <i>today-2</i> , the day before yesterday.
last	Can be used to represent the last date contained in the log file. The parameter <i>last-days</i> specifies the number of days prior to the last date in the log file.
first	Can be used to represent the first date contained in the log file. The parameter <i>first+days</i> specifies the number of days after the first date in the log file.

How to Use It

The stop command is useful if you have a very large log file and do not want to scan the entire file. You can also use it to specify a specific time period for which data is scanned. For example, you can scan a log file for seven-days worth of data that was logged starting a month before the present date.

If you are not sure whether native language support is installed on your system, you can force **utility** to use the C date and time formats by issuing the following statement before running **utility**:

```
LANG=C; export LANG
```

or in C Shell

```
setenv LANG C
```

Example

The scan of 14 days worth of data starts with records logged from July 5, 1999 at 8:00 AM and stops at the last record logged July 18, 1999 at 11:59 pm.

```
utility>  
logfile /var/opt/perf/datafiles/logglob  
detail on  
start 7/5/99 8:00 am  
stop 7/18/99 11:59 pm  
scan
```

To perform the above task using command line arguments, enter:

```
utility -D -b 7/5/99 8:00 am -e 7/18/99 11:59pm -xs
```

Chapter 7: Using the Extract Program

The `extract` program has two main functions: it lets you extract data from raw log files and write it to extracted log files. `Extract` also lets you export log file data for use by analysis products such as spreadsheets.

Note: After the initial installation of the HP Operations agent, services must be started for file installation to complete, before `extract` will function.

The `extract` and `export` functions *copy* data from a log file; *no* data is removed.

Three types of log files are used by the Performance Collection Component:

- `scope` log files, which contain data collected in Performance Collection Component by the `scope` collector.
- extracted log files, which contain data extracted from raw `scope` log files.
- DSI (data source integration) log files, which contain user-defined data collected by external sources such as applications and databases. The data is subsequently logged by Performance Collection Component's DSI programs.

Use the `extract` program to perform the following tasks:

- Extract subsets of data from raw `scope` log files into an extracted log file format that is suitable for placing in archives, for transport between systems, and for analysis by HP Performance Manager. Data *cannot* be extracted from DSI log files.
If you roll back to the `extract` program
- Manage archived log file data by extracting or exporting data from extracted format files, appending data to existing extracted log files, and organizing data by type, date, and shift (hour of day).
- Export data from raw or extracted `scope` log files and DSI log files into ASCII, binary, datafile, or WK1 (spreadsheet) formats suitable for reporting and analysis or for importing into spreadsheets or similar analysis packages.

Note:

1. The `extract` function cannot produce summarized data. Summary data can only be produced by the `export` function.
2. The maximum limits for the `extract` and `export` functions are as follows:
 - Output file from the `extract` function can not exceed 3.5 GB
 - Output file from the `export` function can not exceed 4GB.

Examples of how various tasks are performed and how `extract` commands are used can be found in online help for the `extract` program.

This chapter covers the following topics:

[Running the Extract Program](#)

[Using the Interactive Mode](#)

[Extract Command Line Interface](#)

[Overview of the Export Function](#)

Running the Extract Program

There are three ways to run the `extract` program:

Syntax

The syntax for the command line interface is similar to standard UNIX command line interfaces on other programs and is described in detail in this chapter.

For interactive and batch mode the command syntax is the same: a command followed by one or more parameters. Commands can be entered in any order; if a command has a parameter associated with it, the parameter must be entered immediately after the corresponding command.

There are two types of parameters - *required* (for which there are no defaults) and *optional* (for which defaults are provided). How the `extract` program handles these parameters depends on the mode in which it is running.

In interactive mode, if an *optional* parameter is missing, the program displays the default parameter and lets you either confirm it or override it. If a *required* parameter is missing, the program prompts you to enter the parameter.

In batch mode, if an *optional* parameter is missing, the program uses the default values. If a *required* parameter is missing, the program terminates.

Errors and missing data are handled differently for interactive mode than for command line and batch mode, because you can supply additional data or correct mistakes in interactive mode, but not in command line and batch mode.

Using Interactive Mode

Using the `extract` program's interactive mode requires you to issue a series of commands to execute a specific task.

For example, if you want to export application data collected starting May 15, 2003, from the default global log file, you issue the following commands after invoking the `extract` program

```
logfile /var/opt/perf/datafiles/logglob
application detail
start 5/15/2003
export
```

The `logfile` command opens `/var/opt/perf/datafiles/logglob`, the default global log file. The `start` command specifies that only data logged after 5/15/03 will be exported. The `export` command starts the exporting of the data.

Extract Command Line Interface

In addition to the interactive and batch mode command syntax, command options and arguments can be passed to the `extract` program through the command line interface. The command line interface fits into the typical UNIX environment by allowing the `extract` program to be easily invoked by shell scripts and allowing its input and output to be redirected into UNIX pipes.

For example, the command line equivalent of the example shown in the previous section [Using Interactive Mode](#) is:

```
extract -l -a -b 5/15/02 -xp
```

In command line mode, the global log file `/var/opt/perf/datafiles/logglob` is the default; you do not have to specify it.

Command line options and arguments are listed in the following table. The referenced command descriptions can be found in [Extract Commands](#).

Table 6: Command Line Arguments

Command Option	Argument		Description
-b	date	time	Specifies starting date and time of an extract or export function. (See start command)
-B		UNIXstart time	Specifies starting time in UNIX format for an extract or export function.
-e	date	time	Specifies ending date and time of an extract or export function. (See stop command)
-E		UNIXstop time	Specifies stopping time in UNIX format for an extract or export function.
-s	time-time	noweekends	Specifies start and end time for specific periods excluding weekends. (See shift command)
-l	logfile		Specifies input log file. (See logfile command in Chapter 7.) <code>/var/opt/perf/datafiles/logglob</code> is the default.
-r	export template file		Specifies an export template file for export function. (See report command)
-C	classname	opt	Specifies scope data to extract or export, or self-describing (DSI) data to export. (See class command) opt = detail (default) summary both off
-i			Specifies scope data to extract or export data from logical systems.

Table 6: Command Line Arguments, continued

Command Option	Argument	Description
-k		Exports killed processes only. If you use this option, include the PROC_INTEREST metric in reptfile.
gapkcdzntuy iGADZNTUYI		<p>Specifies types of data to extract/export:</p> <p>g = global detail. (See global command) global detail is off by default.</p> <p>a = application detail. (See application command)</p> <p>p = process detail (See process command)</p> <p>k = process killed. (See process command)</p> <p>c = configuration detail (See configuration command)</p> <p>d= disk device detail (See disk command)</p> <p>z = lvolume detail (See lvolume command)</p> <p>n = netif detail (See netif command)</p>
gapkcdzntuy iGADZNTUYI (continued)		<p>t= transaction detail</p> <p>u = CPU detail</p> <p>y = filesystem detail</p> <p>i = logical systems detail</p> <p>Note: The following summary options are for export only; the extract function does not support data summarization.</p> <p>G = global summary Global summary is off by default.</p> <p>A = application summary</p> <p>D = disk device summary (See disk command)</p> <p>Z=lvolume summary (See lvolume command)</p> <p>N = netif summary (See netif command)</p> <p>I = logical systems summary</p>

Table 6: Command Line Arguments, continued

Command Option	Argument		Description
gapkcdzntuy GADZNTUY (continued)			T = transaction summary U = CPU summary (See cpu command) Y = filesystem summary (See filesystem command)
-v			Generates verbose output report formats.
-f	filename	,new ,append ,purge	Sends extract or export data to a file. If no filename, sends data to default output files. (See output command)
-ut			Shows date and time in UNIX format in exported DSI log file data.
-we	1.....7		Specifies days to exclude from export; 1=Sunday. (See weekdays command description.)
-xp	xopt		Exports data to external format files. (See export command)
-xt	xopt		Extracts data in system internal format. (See extract command) xopt = <i>dwmy</i> (Day Week Month Year) <i>dwmy</i> -[offset] <i>dwmy</i> [absolute]
-xw	week		Extracts a calendar week's data. (See weekly command)
-xm	month		Extracts a calendar month's data. (See monthly command)
-xy	year		Extracts a calendar year's data. (See monthly command)
-? or ?			Displays command line syntax.

When you are evaluating arguments and entering command options on the command line, the following rules apply:

- Errors and missing data are handled exactly as in the corresponding batch mode command. That is, missing data will be defaulted if possible and all errors cause the program to terminate immediately.
- Echoing of commands and command results is disabled unless the -v argument is used to enable verbose mode.

- If no valid action is specified (-xp, -xw, -xm, -xy, or -xt), extract starts reading commands from its stdin file after all parameters have been processed.
- If an action is specified (-xp, -xw, -xm, -xy, or -xt), the program will execute those command options after all other parameters are evaluated, regardless of where they were positioned in the list of parameters.
- If an action is specified in the command line, the extract program will not read from its stdin file; instead it will terminate following the action:
extract -f rxdata -r /var/opt/perf/rept1 -xp d-1 -G

Which translates into:

-f rxdata	Outputs to a file named rxdata in current directory
-r rept1	File /var/opt/perf/rept1 contains the desired export format
-xp d-1	Exports data for this day minus 1 (yesterday)
-G	Exports global summary data.

Note that the actual exporting is not done until the end so the -G parameter is processed before the export is done.

Also notice that the log file was not specified so it uses the default logglob file.

Because an action was specified (-xp), once the export is finished the extract program terminates without reading from its stdin file. In addition, verbose mode was not set with the -v command option so all extraneous output to stdout is eliminated.

Overview of the Export Function

The extract program's export command converts Performance Collection Component raw, extracted, or DSI log file data into exported files. The export command writes files in any one of four possible formats: ASCII, datafile, binary, and WK1 (spreadsheet). Exported files can be used in a variety of ways, such as reports, custom graphics packages, databases, and user-written analysis programs.

How to Export Data

In the simplest form, you can export data by:

- specifying the default global log file, /var/opt/perf/datafiles/logglob, from which you want to export data
- specifying the default export template file, /var/opt/perf/reptfile, that defines the format of the exported data
- starting the export function.

The exported data is placed in a default output file named xfrdGLOBAL.asc in your current directory. The output file's ASCII format is suitable for printing.

If you want to export something other than this default set of data, you can use other commands and files in conjunction with the `export` command.

You can export the following types of data:

global	5-minute and hourly summaries
application	5-minute and hourly summaries
process	One-minute details
disk device	5-minute and hourly summaries
lvolume	5-minute and hourly summaries
transaction	5-minute and hourly summaries
configuration	One record containing parm file information, and system configuration information, for each time the data collector started.
any DSI class	Intervals and summaries for DSI log files
netif	5-minute and hourly summaries
cpu	5-minute and hourly summaries
filesystem	5-minute and hourly summaries

- You can specify which data items (metrics) are needed for each type of data.
- You can specify starting and ending dates for the time period in which the data was collected along with shift and weekend exclusion filters.
- You can specify the desired format for the exported data in an export template file. This file can be created using any text editor or word processor that lets you save a file in ASCII (text) format.
- You can also use the default export template file, `/var/opt/perf/reptfile`. This file specifies the following output format settings:
 - ASCII file format
 - a 0 (zero) for the missing value
 - a blank space as the field separator
 - 60-minute summaries
 - column headings are included
 - a recommended set of metrics for a given data type is included in the export

Note: If you extracting or exporting data from log files which are created from a specific platform, it is recommended that you use the `reptall` file from the same platform. This is because the list of metric classes supported are different on different platforms.

Sample Export Tasks

Two sample export template files, `repthist` and `reptall`, are furnished with Performance Collection Component. These files are located in the `/var/opt/perf/` directory. You can use `repthist` and `reptall` to perform common export tasks or as a starting point for custom tasks, such as the task described next.

Generating a Printable CPU Report

The `repthist` export template file contains the specifications to generate a character graph of CPU and disk usage for the system over time. This graph consists of printable characters that can be printed on any device capable of 132 column printing. For example, you could use the following `extract` program commands to generate a graph of the last seven days and should produce approximately two pages (34 pages if 5-minute detail is specified instead of hourly summaries).

```
logfile /var/opt/perf/datafiles/logglob
report /var/opt/perf/repthist
global summary
start today-7
export
```

The exported data is in an export file named `xfrsGLOBAL.asc`. To print it, type:

```
lp xfrsGLOBAL.asc
```

Producing a Customized Export File

If you want to create a totally new export template file, copy the export template file and customize it using the `extract` program's `guide` command. In guided mode, you copy the `reptall` file from the `/var/opt/perf/` directory and read the `scope` or DSI log file specified to dynamically create the list of data types and metric names.

The `reptall` file contains every possible metric for each type of `scope` log file data so you need only uncomment those metrics that are of interest to you. This is easier than retyping the entire export template file.

Export Data Files

If you used the `output` command to specify the name of an output file prior to issuing the `export` command, all exported data will be written to this single file. If you are running the `extract` program interactively and want to export data directly to your workstation (standard output file), specify the `extract` command output `stdout` prior to issuing the `export` command.

If the output file is set to the default, the exported data is separated into as many as 14 different default output files depending on the type of data being exported.

The default export log file names are:

xfrdGLOBAL.ext	Global detail data file
xfrsGLOBAL.ext	Global hourly summary data file
xfrdAPPLICATION.ext	Application detail data file
xfrsAPPLICATION.ext	Application hourly summary data file
xfrdPROCESS.ext	Process detail data file
xfrdDISK.ext	Disk device detail data file
xfrsDISK.ext	Disk device hourly summary data file
xfrdVOLUME.ext	Logical volume detail data file
xfrsVOLUME.ext	Logical volume summary data file
xfrdNETIF.ext	Netif detail data file
xfrsNETIF.ext	Netif summary detail data file
xfrdCPU.ext	CPU detail data file
xfrsCPU.ext	CPU summary data file
xfrdFILESYSTEM.ext	Filesystem detail data file
xfrsFILESYSTEM.ext	Filesystem summary data file
xfrdTRANSACTION.ext	Transaction detail data file
xfrsTRANSACTION.ext	Transaction summary data file
xfrdCONFIGURATION.ext	Configuration data file

where ext= asc (ASCII), bin (binary), dat (datafile), or wk1 (spreadsheet).

Note: No output file is created *unless* you specify the type and associated items that match the data in the export template file prior to issuing the export command.

Export Template File Syntax

The export template file can contain all or some of the following information, depending on how you want your exported data to be formatted and what you want the export file to contain:

```
report      "export file title"
format      [ASCII]
            [datafile]
            [binary]
            [WK1] or
            [spreadsheet]
```

```

headings      [on]
              [off]
separator= "char"
summary=value
missing=value
layout=single | multiple
output=filename
data type datatype
items
  
```

Parameters

report	Specifies the title for the export file. For more information, see the following section, Export File Title .
format	Specifies the format for the exported data.
	ASCII
	ASCII (or text) format is best for copying files to a printer or terminal. It does not enclose fields with double quotes (").
	Datafile
	The datafile format is similar to ASCII format except that non-numerical fields are enclosed in double quotes. Because double quotes prevent strict column alignment, files in datafile format are not recommended for direct printing. The datafile format is the easiest format to import into most spreadsheets and graphics packages.
	Binary
	The binary format is more compact because numerical values are represented as binary integers. It is the most suitable format for input into user-written analysis programs because it needs the least conversion, and it maintains the highest metric accuracy. It is not suitable for direct printing.
	WK1 (spreadsheet)
	The WK1 (spreadsheet) format is compatible with Microsoft Excel and other spreadsheet and graphics programs.
headings	Specifies whether or not to include column headings for the metrics listed in the export file. If <code>headings off</code> is specified, no column headings are written to the file. The first record in the file is exported data. If <code>headings on</code> is specified, ASCII and datafile formats place the export title plus column headings for each column of metrics written <i>before</i> the first data records. Column headings in binary format files contain the description of the metrics in the file. WK1 formats always contain column headings.
separator	Specifies the character that is printed between each field in ASCII or datafile

	formatted data. The default separator character is a blank space. Many programs prefer a comma as the field separator. You can specify the separator as any printing or nonprinting character.
summary	Specifies the number of minutes for each summary interval. The value determines how much time is included in each record for summary records. The default interval is 60 minutes. The summary value can be set between 5 and 1440 minutes (1 day).
missing	Specifies the data value to be used in place of missing data. The default value for missing data is zero. You can specify another value in order to differentiate missing from zero data. A data item may be missing if it was: <ul style="list-style-type: none"> • not logged by a particular version of the scope collector • not logged by scope because the instance (application, disk, transaction, netif) it belongs to was not active during the interval, or • in the case of DSI log files, no data was provided to the dsilog program during an interval, resulting in “missing records”. Missing records are, by default, excluded from exported data.
layout	Specifies either single or multiple layouts (output per record output) for data types such as application, disk, transaction, lvolume, or netif. Single layout writes one instance per record, for every application that was active during the time interval. Example: One disk exported in one record. Multiple layout writes multiple instances in one record for each time interval, with part of that record reserved for each configured application. Example: All disks exported in one record.
output	Specifies the name of the output file where the exported data will be written. output can be specified for each class or data type exported by placing output <i>filename</i> just after the line indicating the data type that starts the list of exported data items. Any valid file name can be specified for output. You can also specify the name interactively using the output command. Any name you specify overrides the default output file name.
data type	Specifies one of the exportable data types: global, application, process, disk, transaction, lvolume, netif, configuration, or DSI class name. This starts a section of the export template file that lists the data items to be copied when this type of data is exported.
items	Specifies the metrics to be included in the exported file. Metric names are listed, one per line, in the order you want them listed in the resulting file. You must specify the proper data type before listing items. The same export template file can include item lists for as many data types as you want. Each data type will be referenced <i>only</i> if you choose to export that type of data.

The output and layout parameters can be used more than once within an export template file. For example:

```
data type global
  output=myglobal
  gbl_cpu_total_util
```

```
data type application
  output=myapp
  layout=multiple
  app_cpu_total_util
```

You can have more than one export template file on your system. Each one can define a set of exported file formats to suit a particular need. You use the `report` command to specify the export template file to be used with the export function.

Note: You cannot specify different layouts within a single data type. For example, you cannot specify data type `disk` once with `layout = multiple` and again with `layout = single` within the same export file.

Export File Title

The following items can be substituted in the `export file title` string:

<code>!date</code>	The date the export function was performed.
<code>!time</code>	The time the export function was performed.
<code>!logfile</code>	The fully qualified name of the source log file.
<code>!class</code>	The type of data requested.
<code>!collector</code>	The name and version of the collector program. (Not valid with DSI log files.)
<code>!system_id</code>	The identifier of the system that collected the data. (Not valid with DSI log files.)

For example, the string

```
report "!system_id data from !logfile on !date !time"
```

generates an export file title similar to

```
barkley data from logglob on 02/02/99 08:30 AM
```

Creating a Custom Graph or Report

Suppose you want to create a custom graph or report containing exported global and application data. You would do the following:

1. Determine which data items (metrics) are needed from each data type and in what format you will access them.
For this example, you want an ASCII file without headings and with fields separated by commas.
2. Create and save the following ASCII export template file in the `/var/opt/perf/` directory. Name the file `report1`.

```
REPORT "sample export template file (report1)"
FORMAT ASCII
HEADINGS OFF
```

```
DATA TYPE GLOBAL
  GBL_CPU_TOTAL_UTIL
  GBL_DISK_PHYS_IO_RATE
```

```
DATA TYPE APPLICATION
  APP_CPU_TOTAL_UTIL
  APP_DISK_PHYS_IO_RATE
  APP_ALIVE_PROCESSES
```

3. Run the `extract` program.
4. Issue the `report` command to specify the export template file you created.

```
report /var/opt/perf/report1
```

5. Specify global summary data and application summary data using the `global` and `application` commands.

```
global summary application summary
```

6. Issue the `export` command to start the export.

```
export
```

7. Because you did not specify where the program should get the performance data from, you are prompted to do so. In this example, since the default log file is correct, just press `Enter`.

8. The output looks like this:

```
exporting global data .....50%.....100%
exporting application data .....50%.....100%
```

The exported file contains 31 days of data from 01/01/99 to 01/31/99

data type	examined records	exported records	space
global summaries		736	0.20 Mb
application summaries		2560	0.71 Mb
			0.91 Mb

The two files you have just created — `xfrsGLOBAL.asc` and `xfrsAPPLICATION.asc` — contain the global and application summary data in the specified format.

Output of Exported Files

The contents of each exported file are:

export title line	If <code>export title</code> and <code>headings on</code> were specified.
Names (application, netif, lvolume, or transaction)	If <code>headings on</code> was specified along with a multiple layout file.

Heading line1	If headings on was specified.
Heading line2	If headings on was specified.
first data record	
second data record	
...	
last data record	

Report title and heading lines are not repeated in the file.

Notes on ASCII and Datafile Formats

The data in these format files is printable ASCII format. ASCII and datafile formats are identical except that in the latter, all non-numeric fields are enclosed with double quotes. Even the datafile header information is enclosed with double quotes.

The ASCII file format does not enclose fields with double quotes. Therefore, the data in ASCII files will be properly aligned when printed.

Numerical values are formatted based on their range and internal accuracy. Since all fields will not be the same length, be sure to specify the separator you want to use to start each field.

The user-specified separator character (or the default blank space) separates the individual fields in ASCII and datafile formats. Blank spaces, used as separators, can be visually more attractive if you plan to print the report. Other characters can be more useful as separators if you plan to read the export template file with another program.

Using the comma as a separator is acceptable to many applications, but some data items may contain commas *that are not separators*. These commas can confuse analysis programs. The date and time formats can contain different special characters based on the native language specified when you execute the extract program.

Note: To use a nonprinting special character as a separator, enter it into your export template file immediately following the first double quote in the separator parameter.

Tip:

- Most spreadsheets accept files in datafile format using separator=",".
- Many spreadsheet packages accept a maximum of 256 columns in a single sheet. Use care when exporting multiple layout types of data because it is easy to generate more than 256 total items. You can use the output of the report reportfile, show command to determine if you are likely to see this problem.
- If you have a printer that supports underlining, you can create a more attractive printout by specifying ASCII format and the vertical bar character (separator=|) and then printing the file with underlining turned on.

Notes on Binary Format

In binary format files, numerical values are written as 32-bit integers. This can save space by reducing the overall file size, but your program must be able to read binary files. We do not recommend copying a binary format file to a printer or a terminal.

In binary format, non-numerical data is written the same as it was in the ASCII format except separator characters are not used. To properly use a binary format file, you should use the record layout report printed by `extract` when you specify `report reportfile, show`. This report gives you the starting byte for each item specified.

To maintain maximum precision and avoid nonstandard, binary floating-point representations, all numerical values are written as scaled, 32-bit integers. Some items might be multiplied by a constant before they are truncated into integer format.

For example, the number of seconds the CPU was used is multiplied by 1000 before being truncated. To convert the value in the exported file back to the actual number of seconds, divide it by 1000. For ease in conversion, specify `headings on` to write the scale factors to the exported file. The report title and special header records are written to binary format files to assist in programmatic interpretation.

Binary integers are written in the format that is native to the system on which the `extract` program is being run. For example, Intel systems write “little endian” integers while HP-UX, IBM AIX, and Sun systems write “big endian” integers. Use care when transporting binary exported files to systems that use different “endians”.

Binary Header Record Layout

Each record in a binary format exported file contains a special 16-byte record header preceding any user-specified data. The `report reportfile, show` command includes the following four fields that make up this record header:

Binary Title Record

The Title Record for BINARY files contains information designed to assist programmatic interpretation of the exported file's contents. This record will be written to the exported file whenever `headings on` is specified.

The contents of the Binary Title Record are:

Record Length	4 byte Int	Length of Title Record
Record ID	4 byte Int	-1
Date_Seconds	4 byte Int	Date exported file was created
Number_of_vars	4 byte Int	Maximum number of repeating variables
Size of Fixed Area	4 byte Int	Bytes in nonvariable part of record

Size of Variable Entry	4 byte Int	Bytes in each variable entry
GMT Time Offset	4 byte Int	Seconds offset from Greenwich Mean Time
Daylight Savings Time	4 byte Int	=1 indicates Daylight Savings Time
System ID	40 Characters,	System Identification
Collector Version	16 Characters,	Name & version of the data collector
Log File Name	72 Characters,	Name of the source log file
Report Title	100 Characters,	User specified report title

The Date_Seconds, GMT Time Offset, and Daylight Savings Time metrics in the Binary Title Record apply to the system and time when the export file was created. If this is not the same system that logged the data, these fields cannot properly reflect the data in the file.

Binary Item Identification Record

The first header record (record ID -2) in the binary file contains the unique item numbers for each item exported. Each Item ID is a 4-byte integer number that can be cross referenced using the `rxitemid` file supplied with this product. The Item ID fields are aligned with the data fields they represent in the rest of the file. All binary exported data items will occupy a multiple of 4 bytes in the exported file and each will start on a 4-byte boundary. If a data item requires more than 4 bytes of space, its corresponding item ID field will be zero filled on the left.

For example, the process metric Program requires 16 bytes. Its data and item ID records would be:

```
Header 1 (Item Id Record) ...| 0| 0| 0|12012|
Process Data Record      |Prog|ram_|name|_aaa|
```

Binary Scale Factor Record

The second header record (record ID -3) in the binary file contains the scale factors for each of the exported items. Numeric items are exported to binary files as 32-bit (4-byte) integers in order to minimize problems with the way in which different computer architectures implement floating point. Before being truncated to fit into the integer format, most items are multiplied by a fixed scale factor. This allows floating point numbers to be expressed as a fraction, using the scale factor as a denominator.

Each scale factor is a 32-bit (4-byte) integer to match the majority of data items. Special values for the scale factors are used to indicate non-numeric and other special valued metrics.

Special Scale Factors

Non-numeric metrics, such as ASCII fields, have zero scale factors. A negative 1 scale factor should not occur, but if it does it indicates an internal error in the extract program and should be reported.

The DATE format is MPE CALENDAR format in the least significant 16 bits of the field (the 16 bits farthest right). The scale factor for date is 512. Scaling this as a 32-bit integer (dividing by 512) isolates the year as the integer part of the date and the day of the year (divided by 512) as the fractional part.

TIME is a 4-byte binary field (hour, minute, second, tenths of seconds). The scale factor for time is 65536. Dividing it by 65536 forms a number where the integer part is the (hour * 256) + minute.

It is easier to handle a Date_Seconds value in a binary file.

Special Scale Factors

Non-numeric metrics, such as ASCII fields, have zero scale factors. A negative 1 scale factor should not occur, but if it does it indicates an internal error in the extract program and should be reported.

The DATE format is MPE CALENDAR format in the least significant 16 bits of the field (the 16 bits farthest right). The scale factor for date is 512. Scaling this as a 32-bit integer (dividing by 512) isolates the year as the integer part of the date and the day of the year (divided by 512) as the fractional part.

TIME is a 4-byte binary field (hour, minute, second, tenths of seconds). The scale factor for time is 65536. Dividing it by 65536 forms a number where the integer part is the (hour * 256) + minute.

It is easier to handle a Date_Seconds value in a binary file.

Transaction Name Record

When transaction data is exported in the Multiple Layout format, a special Transaction Name Record is written to identify the application-transaction name. For binary format files, this record has a record ID - 5. It consists of the binary record 16-byte header and a 60-byte truncated application-transaction name for each transaction that was configured at the starting date of the exported data. If transactions are added during the time covered in the data extraction, the Transaction Name Record will be repeated with the new application-transaction name appended to the end of the original list. Transactions that are deleted after the start of the data extraction will not be removed from the Multiple Layout data record.

Disk Device Name Record

When disk device data is exported in the Multiple Layout format, a special Disk Device Name Record is written to identify the disk device name. For binary format files, this record has a record ID -7. It consists of the binary record 16-byte header and a 20-byte disk device name for each disk device that was configured at the starting date of the exported data.

If disk devices are added during the time covered in the data extraction, the Disk Device Name Record will be repeated with the new disk device name appended to the end of the original list. Disk devices that are deleted after the start of the data extraction will not be removed from the Multiple Layout data record.

Logical Volume Name Record

When logical volume data is exported in the Multiple Layout format, a special Logical Volume Name Record is written to identify the logical volume name. For binary format files, this record has a record ID -8. It consists of the binary record 16-byte header and a 20-byte disk device name for each logical volume that was configured at the starting date of the exported data.

If logical volumes are added during the time covered in the data extraction, the Logical Volume Name Record will be repeated with the new logical volume name appended to the end of the original list. Logical volumes that are deleted after the start of the data extraction will not be removed from the Multiple Layout data record.

Netif Name Record

When `netif` data is exported in the Multiple Layout format, a special Netif Name Record is written to identify the `netif` device name. For binary format files, this record has a record ID -11. It consists of the binary record 16-byte header and a 20-byte `netif` device name for each `netif` device that was configured at the starting date of the exported data.

If `netif` devices are added during the time covered in the data extraction, the Netif Name Record will be repeated with the new device name appended to the end of the original list. Netif devices that are deleted after the start of the data extraction will not be removed from the Multiple Layout data record.

Chapter 8: Extract Commands

This chapter describes the **extract** program's commands. It includes a table showing command syntax, a table of commands for extracting and exporting data, and a command reference section describing the commands in alphabetical order.

Commands and parameters for **extract** can be entered with any combination of uppercase and lowercase letters. Only the first three letters of the command's name are required, *except* for the weekdays and weekly commands that require you to enter the whole name. For example, the command application detail can be abbreviated as app det.

Examples of how these commands are used can be found in online help for the **extract** program.

The table on the following pages summarizes the syntax of the **extract** commands and their parameters.

Note: The extract function cannot produce summarized data. Summary data can only be produced by the export function.

Table 7 Extract Commands: Syntax and Parameters

Command	Parameter
application	on detail summary (export only) both (export only) off (default)
class	detail (default) summary (export only) both (export only) off
cpu	detail summary (export only) both (export only) off (default)
configuration	on detail off (default)
disk	on detail

Table 7 Extract Commands: Syntax and Parameters, continued

Command	Parameter
	summary (export only) both (export only) off (default)
exit e	
export	day[<i>ddd</i>] [- <i>days</i>] week [<i>ww</i>] [- <i>weeks</i>] month[<i>mm</i>] [- <i>months</i>] year [<i>yy</i>] [- <i>years</i>]
extract	day[<i>ddd</i>] [- <i>days</i>] week [<i>ww</i>] [- <i>weeks</i>] month[<i>mm</i>] [- <i>months</i>] year [<i>yy</i>] [- <i>years</i>]
filesystem	detail summary(export only) both (export only) off (default)
global	ondetail (default) summary (export only) both (export only) off
guide	
help	
list	filename *
logfile	logfile
lvolume	on detail summary (export only) both (export only) off (default)

Table 7 Extract Commands: Syntax and Parameters, continued

Command	Parameter
menu	
monthl	
y	<i>yyymm</i> <i>mm</i>
netif	on detail summary (export only) both (export only) off (default)
output	<i>outputfile</i> ,new ,purgeboth ,append
process	on detail [app#[-#],...] off (default) killed
quit q	
report	[<i>export template file</i>], show
shift	<i>starttime - stoptime</i> all day noweekends
sh !	shell command
show	all
start	<i>date[time]</i> today[-days][<i>time</i>] last[-days][<i>time</i>] first[+days][<i>time</i>]
stop	<i>date[time]</i>

Table 7 Extract Commands: Syntax and Parameters, continued

Command	Parameter
	today[-days][time] last[-days][time] first[+days][time]
transaction	on detail summary (export only) both (export only) off (default)
weekdays	1.....7
weekly	yyww ww
yearly	yyyy yy

The following table lists the commands that are used for extracting and exporting data and the types of log files used (scope log files or DSI log files).

Table 8 Extract Commands: Extracting and Exporting Data

Command	Extract Data	Export Data	scope Log Files	DSI Log Files
application	x	x	x	
class	x	x	x	x
configuration		x	x	
cpu	x	x	x	
disk	x	x	x	
export		x	x	x
extract	x		x	
filesystem	x	x	x	
global	x	x	x	
logfile	x	x	x	x
lvolume	x	x	x	

Table 8 Extract Commands: Extracting and Exporting Data, continued

Command	Extract Data	Export Data	scope Log Files	DSI Log Files
monthly	x		x	
netif		x	x	
output	x	x	x	x
process	x	x	x	
report		x	x	x
shift	x		x	x
start	x	x	x	x
stop	x	x	x	x
transaction	x	x	x	x
weekdays		x	x	x
weekly	x		x	
yearly	x		x	
logicalsystems	x	x	x	

application

Use the application command to specify the type of application data that is being extracted or exported.

The default is application off

Syntax

application	[on] [detail] [summary] [both] [off]
--------------------	---

Parameters

on or detail	Specifies that raw, 5-minute detail data should be extracted or exported.
summary (export only)	Specifies that data should be summarized by: the number of minutes specified with the summary parameter in the

	<p>specified export template file (export only)</p> <p>the default summary interval of one hour (export or extract)</p> <p>Summarization can significantly reduce the size of the resulting extracted or exported data, depending on the summarization interval used. For example, hourly summary data is about one-tenth the size of 5-minute detail data.</p>
both (export only)	Specifies that detail data and summary data are to be extracted or exported.
off	Specifies that no data of this type is to be extracted or exported.

Note: If you are using Performance Manager, detail data must be included in an extracted file before drawing application graphs with points every 5 minutes.

Example

In this example, the application command causes detailed application log file data to be exported: The **output export** file contains the application metrics specified in the **myrept** export template file.

```
logfile /var/opt/perf/datafiles/logglob
global off
application detail
report /var/opt/perf/myrept
export
```

To perform the above task using command line arguments, enter:

```
extract -a -r /var/opt/perf/myrept -xp
```

class

Use the class command to specify the class of DSI data to be exported, or scope data to be extracted or exported.

The default is class detail.

Syntax

		[detail]
class	[<i>classname</i>]	[summary]
		[both]
		[off]

Parameters

classname	Name of a group similarly classified metrics.
detail	For DSI log files, specifies how much detail data is exported according to the time set in DSI log file. For more information, see Overview of Data Source Integration . For scope log files, specifies that raw, 5-minute detail should be extracted or exported.
summarybothoff	See “Parameters” in the description of the command application . Summary and both can only be exported.

Examples

To export summary data in a DSI log file that contains a class named `acctg_info`, issue the following command:

```
class acctg_info summary
```

Once the log file is specified by the user and opened by the **extract** program, the `acctg_info` class is verified to exist in the log file and can subsequently be exported.

Other variations of this command are:

```
CLASS ACCTG_INFO SUMMARYclass ACCTG_INFO summaryclass acctg_info sum
```

Commands can be either uppercase or lowercase. Class names are always upshifted and then compared.

In the following example, summary data in a class named `fin_info` is exported.

```
extract>
class fin_info summary
export
```

To perform the above task using command line arguments, enter:

```
extract -l dsi.log -C fin_info summary -xp
```

configuration

Use the configuration command to specify whether or not to export system configuration information.

The default is configuration off.

Syntax

configuration	[on] [detail] [off]
----------------------	----------------------------

Parameters

on or detail	Specifies that all configuration records should be exported.
--------------	--

off	Specifies that no configuration data is to be exported.
-----	---

All configuration information available in the log file is exported. Any begin, end, shift, start, stop or nowweekends commands that are used with the configuration command are ignored.

Note: The configuration command affects only the export function. The extract function is not affected because it always extracts system configuration information.

Example

In this example, the configuration command causes system configuration information to be exported. The output export file contains the configuration metrics specified in the **myrept** export template file.

```
logfile /var/opt/perf/datafiles/logglob  
configuration on  
report /var/opt/perf/myrept  
export
```

To perform the above task using command line arguments, enter:

```
extract -c -r /var/opt/perf/myrept -xp
```

cpu

Use the cpu command to specify the summarization level of CPU.

The default is cpu off.

Syntax

cpu	[detail]
	[summary] [both]
	[off]

Parameters

detail	Extracts or exports 5-minute detail records.
summary	Exports summary records.
both	Exports both detail and summary records.
off	Extracts or exports no CPU data.

Example

In this example, the cpu command causes CPU detail data that was collected starting July 26, 2001 to be exported. Because no export template file is specified, the default export template file, **reptfile**, is used. All disk metrics are included in the output file as specified by **reptfile**.

```
logfile /var/opt/perf/datafiles/logglob
```

```
global off  
cpu detail  
start 7/26/01  
export
```

To perform the above task using command line arguments, enter:

```
extract -u -b 7/26/01 -xp
```

disk

Use the disk command to specify the type of disk device data that is being extracted or exported.

The default is disk off .

Syntax

disk	[on][detail][summary][both][off]
-------------	--

Parameters

on or detailsummaryboth off	See "Parameters" in the description of the "application" on page 161 command at the beginning of this chapter. Summary and both can only be exported.
-----------------------------------	---

Example

In this example, the disk command causes disk detail data that was collected starting July 5, 1999 to be exported. Because no export template file is specified, the default export template file, **reptfile**, is used. All disk metrics are included in the output file as specified by **reptfile**.

```
logfile /var/opt/perf/datafiles/logglob  
global off  
disk detail  
start 7/5/99  
export
```

To perform the above task using command line arguments, enter:

```
extract -D -b 7/5/99 -xp
```

exit

Use the exit command to terminate the **extract** program. The `exit` command is equivalent to the **extract** program's quit command.

Syntax

```
exit
```

```
e
```

export

Use the export command to start the process of copying data into an exported file format.

Syntax

export	[day [week month [year	[ddd] [yyddd] [-days]][ww] [yyww] [-weeks]][mm] [yymm] [-months]][yy] [yyyy] [-years]]
---------------	-----------------------------------	--

Parameters

Use one of the following parameters to export data for a particular interval.

day	Represents a single day
week	Represents a single week, Monday through Sunday
month	Represents a single month, first through last calendar day
year	Represents a single year, first through last calendar day

If no parameters are used with the export command, the interval used for the exported data is set by the start and stop commands.

How to Use It

There are four ways to specify a particular interval (day, week, month, year).

- Current interval - Specify the parameter only. For example, month means the current month.
- Previous interval - Specify the parameter, a minus, and the number of intervals before the current one desired. For example, day-1 is yesterday, week-2 is two weeks prior to the current week.
- Absolute interval - Specify the parameter and a positive number. The number indicates the absolute interval desired in the current year. For example, day 2 is January 2 of the current year.

- Absolute interval plus year - Specify the parameter and a large positive number. The number should consist of the last two digits of the year and the absolute interval number in that year. In this format the absolute day would have 5 digits (99002 means January 2, 1999) and all other parameters would have four digits (month 9904 means April of 1999).

If you have not previously specified a log file or an export template file, the logfile command uses the default global log file **logglob** and the report command uses the default export template **filereptfile**.

The settings or defaults for all other parameters are used. For details on their actions, see descriptions of the application, configuration, global, process, disk, lvolume, netif, CPU, filesystem, transaction, output, shift, start, and stop commands.

The export command creates up to 16 different default output files based on the types of data and level of summarization specified.

xfrdGLOBAL.ext	Global detail data file
xfrsGLOBAL.ext	Global hourly summary data file
xfrdAPPLICATION.ext	Application detail data file
xfrsAPPLICATION.ext	Application hourly summary data file
xfrdPROCESS.ext	Process detail data file
xfrdDISK.ext	Disk device detail data file
xfrsDISK.ext	Disk device summary data file
xfrdVOLUME.ext	Logical volume detail data file
xfrsVOLUME.ext	Logical volume summary data file
xfrdNETIF.ext	Netif detail data file
xfrsNETIF.ext	Netif summary data file
xfrdCPU.ext	CPU detail data type
xfrsCPU.ext	CPU summary data type
xfrdFILESYSTEM.ext	Filesystem detail data type
xfrsFILESYSTEM.ext	Filesystem summary data type
xfrdTRANSACTION.ext	Transaction detail data file
xfrsTRANSACTION.ext	Transaction summary data file
xfrdCONFIGURATION.ext	Configuration detail data file

where **ext** = **asc**, **dat**, **bin**, or **wk1**

The default file names are created from the data type name. The prefix is either **xfrd** or **xfrs** depending if the data is detailed or summary data. The extension is the specified **asc** (ASCII), **bin** (binary), **dat** (datafile), or **wk1** (spreadsheet) data format.

For example, classname = ACCTG_INFO would have **export** file names of:

<code>xfrdACCTG_INFO.wk1</code>	detailed spreadsheet data for ACCT_INFO
<code>xfrsACCTG_INFO.asc</code>	summarized ASCII data for ACCT_INFO

For more information about exporting data, see [Overview of the Export Function](#).

Example

In this example, the export command causes log file data collected yesterday from 8:00 am to 5 pm to be exported. Because no export template file is specified, the default export template file, **reptfile**, is used. All global metrics are included in the output file as specified by **reptfile**

```
logfile /var/opt/perf/datafiles/logglob
```

```
start today-1 8:00 am
```

```
stop today-1 5:00 pm
```

```
global both
```

```
export
```

To perform the above task using command line arguments, enter:

```
extract -gG -b today-1 8:00 am -e today-1 5:00 pm -xp
```

extract

Use the `extract` command to start the process of copying data from raw log files into an extracted file format. Extracted files can be used for archiving or for analysis by analyzer programs such as Performance Manager. You can extract data from raw log files and from extracted files.

The `extract` command cannot be used to process data from DSI log files.

Syntax

extract	[day [week [month [year	[ddd] [yyddd] [-days]][ww] [yyww] [-weeks]][mm] [yymm] [-months]][yy] [yyyy] [-years]]
----------------	--------------------------------	--

Parameters

Use one of the following parameters to extract data for a particular interval:

day	Represents a single day
week	Represents a single week, Monday through Sunday
month	Represents a single month, first through last calendar day
year	Represents a single year, first through last calendar day

If no parameters are used with the `extract` command, the interval used for data extraction is set by the start and stop commands.

How to Use It

There are four ways to specify a particular interval (day, week, month, year).

- Current interval - Specify the parameter only. For example, month means the current month.
- Previous interval - Specify the parameter, a minus, and the number of intervals before the current one desired. For example, day-1 is yesterday, week-2 is two weeks prior to the current week.
- Absolute interval - Specify the parameter and a positive number. The number indicates the absolute interval desired in the current year. For example, day 2 is January 2 of the current year.
- Absolute interval plus year - Specify the parameter and a large positive number. The number should consist of the last two digits of the year and the absolute interval number in that year. In this format, the absolute day would have five digits (99002 means January 2, 1999) and all other parameters would have four digits (month 99904 means April of 1999).

The `extract` command starts data extraction. If not previously specified, the logfile and output commands assume the following defaults when the `extract` command is executed:

```
log file = /var/opt/perf/datafiles/logglob
```

```
output file = rxlog,new
```

The settings or defaults for all other parameters are used. For details on their actions, see descriptions of the application, global, process, disk, lvolume, netif, CPU, filesystem, transaction, shift, start, and stop commands.

The size of an extracted log file cannot exceed 3.5 gigabytes.

Example

In the first example, data collected from March 1, 2000 to June 30, 2000 during the hours 8:00 am to 5:00 pm on weekdays is extracted. Only global and application detail data is extracted.

```
logfile /var/opt/perf/datafiles/logglob
```

```
start 03/01/00
```

```
stop 06/30/00
```

```
shift 8:00 am - 5:00 pm noweekends
```

```
global detail
```

```
application detail
```

```
extract
```

To perform the above task using command line arguments, enter:

```
extract -ga -b 03/01/00 -e 6/30/00 -s 8:00 am - 5:00 noweekends -xt
```

In the second example, a new extracted log file named **rxjan00** is created. Any existing file that has this name is purged. All raw log file data collected from January 1, 2000 through January 31, 2000 is extracted:

```
logfile /var/opt/perf/datafiles/logglob
```

```
output rxjan00,purge
```

```
start 01/01/00  
stop 01/31/00  
global detail  
application detail  
transaction detail  
process detail  
disk detail  
lvolume detail  
netif detail  
filesystem detail  
cpu detail  
extract
```

To perform the above task using command line arguments, enter:

```
extract -f rxjan00,purge -gatpdznyu -b 01/01/00 -e 01/31/00 -xt
```

filesystem

Use this command to specify the summarization level of filesystem data to **extract** or **export**.

The default is filesystem off.

Syntax

filesystem	[detail] [summary] [both] [off]
-------------------	---

Parameters

detail	Extracts or exports 5-minute detail records.
summary	Exports summary records.
both	Exports both detail and summary records.
off	Extracts or exports no filesystem data.

Example

In this example, the filesystem command causes filesystem detail data that was collected starting July 26, 2001 to be exported. Because no export template file is specified, the default export template file, **reptfile**, is used. All filesystem metrics are included in the output file as specified by **reptfile**.

```
logfile /var/opt/perf/datafiles/logglob
```

```
global off  
filesystem detail  
start 7/26/01  
export
```

To perform the above task using command line arguments, enter:

```
extract -y -b 7/26/01 -xp
```

global

Use the global command to specify the amount of global data to be extracted or exported.

The default is global detail. (In command line mode, the default is global off .)

Syntax

global	<code>[on] [detail] [summary] [both] [off]</code>
---------------	---

Parameters

<code>detail or onsummarybothoff</code>	See "Parameters" in the description of the "application" on page 161 command at the beginning of this chapter. Summary and both can only be exported.
---	---

How to Use It

Detail data must be extracted if you want to draw Performance Manager global graphs with points every 5 minutes.

Summarized data is graphed by Performance Manager more quickly since fewer data records are needed to produce a graph. If only global summaries are extracted, Performance Manager global graphs cannot be drawn with data points every 5 minutes.

The both option maintains the access speed gained with the hourly summary records while permitting you to draw Performance Manager global graphs with points every 5 minutes.

The off parameter is not recommended if you are using Performance Manager because you must have global data to properly understand overall system behavior. Performance Manager global graphs cannot be drawn unless the extracted file contains at least one type of global data.

Example

The global command is used here to specify that *no* global data is to be exported (global detail is the default). Only detailed transaction data is exported. The **output export** file contains the transaction metrics specified in the **myrept** export template file.

```
extract>
```

```
logfile /var/opt/perf/datafiles/logglob  
global off  
transaction detail  
report /var/opt/perf/myrept  
export
```

To perform the above task using command line arguments, enter:

```
extract -l -t -r /var/opt/perf/myrept -xp
```

guide

Use the `guide` command to enter guided commands mode. The guided command interface leads you through various `extract` commands and prompts you to perform some of the most common tasks that are available.

Syntax

```
guide
```

How to Use It

To enter guided commands mode from **extract's** interactive mode, type **guide**.

To accept the default value for a parameter, press **Return**.

To terminate guided commands mode and return to interactive mode, type **q** at the `guide>` prompt.

This command does not provide all possible combinations of parameter settings. It selects settings that should produce useful results for the majority of users. You can obtain full control over **extract's** functions through **extract's** interactive command mode.

Note: If you are exporting DSI log file data, we recommend using guided commands mode to create a customized export template file and export the data.

help

Use the `help` command to access online help.

Syntax

```
help [keyword]
```

How to Use It

You can enter parameters to obtain information on **extract** commands and tasks, or on help itself. You can navigate to different topics by entering a key word. If more than one page of information is available, the display pauses and waits for you to press **Return** before continuing. Type **q** or **quit** to exit the help system and return to the **extract** program.

You can also request help on a specific topic. For example,

```
help tasks
```

or

```
help resize parms
```

When you use this form of the help command, you receive the help text for the specified topic and remain in the extract command entry context. Because you do not enter the help subsystem interactively, you do not have to type **quit** before entering the next extract command.

list

Use the list command to specify the list file for all extract program reports.

Syntax

list	[file]
	[*]

How to Use It

You can use list at any time while using **extract** to specify the list device. It uses a file name or list device name to output the user-specified settings. If the list file already exists, the output is appended to it.

The data that is sent to the list device is also displayed on your screen.

While **extract** is running, type:

```
list outfilename
```

To return the listing device to the user terminal, type:

```
list stdout
```

or

```
list *
```

To determine the current list device, type the list command without parameters as follows:

```
list
```

If the list file is not **stdout**, most commands are echoed to the list file as they are entered.

Example

The following example, the list device is set to mylist. The results of the next commands are printed to mylist and displayed on your screen.

```
extract>  
logfile /var/opt/perf/datafiles/logglob  
list mylist  
global detail  
shift 8:00 AM - 5:00 PM  
extract
```

logfile

Use the logfile command to open a log file. You must open a log file for all `extract` program functions. You can do this explicitly by issuing the logfile command, or implicitly by issuing the `extract` command or `export` command. If you do not specify a log file name, the **extract** program prompts you for a log file name and displays the default global log file `/var/opt/perf/datafiles/logglob`. You can either accept the default or specify a different log file.

Syntax

logfile [*logfile*]

How to Use It

To open a log file, you can specify the name of either a raw or extracted log file. You cannot specify the name of a file created by the `export` command. If you specify an extracted log file name, all information is obtained from this single file. If you specify a raw log file name, you must specify the name of the global log file before you can access the raw log file. This is the only raw log file name you should specify.

If the log file is not in your current working directory, you must provide its path.

The global log file and other raw log files must be in the same directory. They have the following names:

logglob	global log file
logappl	application log file
logproc	process log file
logdev	device log file
logtran	transaction log file
logindx	index log file

The general contents of the log file are displayed when the log file is opened.

Note: Do not rename raw log files! When accessing these files, the program assumes that the standard log file names are in effect. If you must rename log files to place log files from multiple systems on the same system for analysis, you should first extract the data and then rename the extracted log files.

Example

This is a typical listing of the default global log file.

```
Global      file: /var/opt/perf/datafiles/logglob, version D
Application file: /var/opt/perf/datafiles/logappl
Process     file: /var/opt/perf/datafiles/logproc
Device      file: /var/opt/perf/datafiles/logdev
```

```
Transaction file: /var/opt/perf/datafiles/logdev
Index      file: /var/opt/perf/datafiles/logindx
System ID:  homer
System Type 9000/715/ S/N 2223334442 O/S HP-UX B.10.20 A
Data collector: SCOPE/UX C.02.30
File Created: 10/08/99
Data Covers: 44 days to 11/20/99
Shift is:    All Day
Data records available are:
Global Application Process Disk Volume Transaction
Maximum file sizes:
Global=10.0 Application=10.0 Process=20.0 Device=10.0 Transaction=10.0 MB
The first GLOBAL      record is on 10/08/99 at 08:17 AM
The first NETIF       record is on 10/08/99 at 08:17 AM
The first APPLICATION record is on 11/17/99 at 12:15 PM
The first PROCESS     record is on 10/08/99 at 08:17 AM
The first DEVICE      record is on 10/31/99 at 10:45 AM
The Transaction data file is empty
The default starting date & time = 10/08/99 11:50 AM (LAST -30)
The default stopping date & time = 11/20/99 11:59 PM (LAST -0)
```

lvolume

Use the `lvolume` command to specify the type of logical volume data that is being extracted or exported. (This command is used only on HP-UX systems.)

The default is `lvolume off`.

Syntax

lvolume	[on] [detail] [summary] [both] [off]
----------------	---

Parameters

on or detail summary both off	See "Parameters" in the description of the "application" on page 161 command at the beginning of this chapter. Summary and both can only be exported.
--	---

Example

In this example, a new extracted log file named **rx899** is created and any existing file that has that name is purged. All logical volume data collected from August 1 through August 31 is extracted.

```
logfile /var/opt/perf/datafiles/logglob  
output rx899,purge  
start 08/01/99  
stop 08/31/99  
global detail  
lvolume detail  
month 9908
```

To perform the above task using command line arguments, enter:

```
extract -f rx899,purge -gz -xm 9908
```

menu

Use the `menu` command to print a list of the available **extract** commands.

Syntax

```
menu
```

Example

Command	Parameters	Function
HELP	[topic]	Get information on commands and options
GUIDE	Enter guided commands mode for novice users	
LOGFILE	[logname]	Specify a log file to be processed

LIST	[filename]*]	Specify the listing file
OUTPUT	[filename] [,NEW/PURGE/APPEND] Specify a destination file	
REPORT	[filename][,SHOW]	Specify an Export Format file for EXPORT"
GLOBAL	[DETAIL/SUMMARY/BOTH/OFF]	Extract GLOBAL records
APPLICATION	[DETAIL/SUMMARY/BOTH/OFF]	Extract APPLICATION records
PROCESS	[DETAIL/OFF/KILLED][APP=]	Extract PROCESS records
DISK	[DETAIL/SUMMARY/BOTH/OFF]	Extract DISK DEVICE records
LVOLUME	[DETAIL/SUMMARY/BOTH/OFF]	Extract Logical VOLUME records
NETIF	[DETAIL/SUMMARY/BOTH/OFF]	Extract Logical NETIF records
CPU	[DETAIL/SUMMARY/BOTH/OFF]	Extract CPU records
FILESYSTEM	[DETAIL/SUMMARY/BOTH/OFF]	Extract FILESYSTEM records
CONFIG	[DETAIL/OFF]	Export CONFIGURATION records
CLASS classname	[DETAIL/SUMMARY/BOTH/OFF]	Export classname records
TRANSACTION	[DETAIL/SUMMARY/BOTH/OFF]	Extract TRANSACTION records
START	[startdate time]	Specify a starting date and time for SCAN
STOP	[stopdate time]	Specify an ending date and time for SCAN
SHIFT	[starttime - stoptime] [NOWEEKENDS]	Specify daily shift times

SHOW	[ALL]	Show the current program settings
EXPORT	[d/w/m/y][-offset]	Copy log file records to HOST format files
EXTRACT	[d/w/m/y][-offset]	Copy selected records to output (or append) file
WEEKLY	[ww/yyww]	Extract one calendar week's data with auto file names
MONTHLY	[mm/yymm]	Extract one calendar month's data with auto file names
YEARLY	[yy/yyyy]	Extract one calendar year's data with auto file names
WEEKDAYS	[1...7]	Set days to exclude from export 1=Sunday ! or SH [command] Execute a system command
MENU or ?		List the command menu (this listing)
EXIT or Q		Terminate the program

monthly

Use the monthly command to specify data extraction based on a calendar month. During execution, this command sets the start and stop dates to the proper dates, based on the month and year of the data extracted.

The name of the output file consists of the letters **rxmo** followed by the four digits of the year and the two-digit number of the month being extracted. For example, data extracted in March 1999 would be output to a file named **rxmo199903**.

Syntax

monthly	[yymm] [mm]
----------------	----------------

Parameters

<code>monthly</code>	Extracts data from the current (default) month.
<code>monthly <i>mm</i></code>	Extracts data for a specific month from the current year's data (where <i>mm</i> is a number from 01 to 12).
<code>monthly<i>yy</i><i>mm</i></code>	Extracts data for a specific month and year (where <i>yy</i> <i>mm</i> is a single number consisting of the last two digits of the year and two-digit month number). For example, to extract data for February 1999, specify monthly 9902 .

If you do not specify the log file before executing the `monthly` command, the default **logglob** file is used.

How to Use It

Use the `monthly` command when you are extracting data for archiving on a monthly basis.

The type of data extracted and summarized follows the normal rules for the `extract` command and can be set before executing the `monthly` command. These settings are honored unless a `monthly` output file already exists. If it does, data is appended to it based on the type of data that was originally specified.

The `monthly` command has a feature that opens the previous month's extracted file and checks to see if it is filled--whether it contains data extracted up to the last day of the month. If not, the `monthly` command appends data to this file to complete the previous month's extraction.

For example, a `monthly` command is executed on May 7, 1999. This creates a log file named **rxmo199905** containing data from May 1 through the current date (May 7).

On June 4, 1999, another `monthly` command is executed. Before the **rxmo199906** file is created for the current month, the **rxmo199905** file from the previous month is opened and checked. When it is found to be incomplete, data is appended to it to complete the extraction through May 31, 1999. Then, the **rxmo199906** file is created to hold data from June 1, 1999 to the current date (June 4).

As long as you execute the `monthly` command at least once a month, this feature will complete each month's file before creating the next month's file. When you see two adjacent `monthly` files--for example, **rxmo199905** (May) and **rxmo199906** (June)--you can assume that the first file is complete for that month and it can be archived and purged.

Note: The `monthly` and `extract month` commands are similar in that they both extract one calendar month's data. The `monthly` command ignores the setting of the output command, using instead predefined output file names. It also attempts to append missing data to the previous month's extracted log file if it is still present on the system. The `extract month` command, on the other hand, uses the settings of the output command. It cannot append data to the previous month's extracted file since it does not know its name.

Example

In this example, detail application data logged during May 1999 is extracted.

```
logfile /var/opt/perf/datafiles/logglob
global off
application detail
monthly 9905
```

To perform the above task using command line arguments, enter:

```
extract -a -xm 9905
```

netif

Use the netif command to specify the type of logical network interface (LAN) data to **extract** or **export**. Netif data is logged in the **logdev** file.

The default is netif off.

Syntax

netif	[on] [detail] [summary] [both] [off]
--------------	--

Parameters

on or detail summary bothoff	See "Parameters" in the description of the "application" on page 161 command at the beginning of this chapter. Summary and both can only be exported.
------------------------------------	---

Example

In this example, netif detail data collected from March 1, 2000 to June 30, 2000 during the hours 8:00 am to 5:00 pm on weekdays is extracted.

```
logfile /var/opt/perf/datafiles/logglob  
start 03/01/00  
stop 06/30/00  
shift 8:00 AM - 5:00 PM noweekends  
netif detail  
extract
```

To perform the above task using command line arguments, enter:

```
extract -n -b 03/01/00 -e 6/30/00 -s 8:00 am - 5:00 noweekends -xt
```

output

Use the output command to specify the name of an output file for the extract or export functions.

The optional second parameter specifies the action to be taken if an output file with the same name exists.

Syntax

output	[<i>filename</i>]	[, new] [, purge] [, append]
---------------	------------------------	--

Parameters

,new	Specifies that the output file must be a new file. This is the default action in batch mode. If a file with the same name exists, the batch job terminates.
,purge	Specifies that any existing file should be purged to make room for the new output file.
,append	Specifies that an existing extracted file should have data appended to it. If no file exists with the output file name specified, a new file is created.

How to Use It

If you do not specify an action in batch mode, the default action, new is used. In interactive mode, you are prompted to enter an action if a duplicate file is found.

If you do not specify an output file, default output files are created. The default output file names are:

For **extract: rxlog**

For **export:**

xfrdGLOBAL.ext
xfrsGLOBAL.ext
xfrdAPPLICATION.ext
xfrsAPPLICATION.ext
xfrdPROCESS.ext
xfrdDISK.ext
xfrsDISK.ext
xfrdLVOLUME.ext
xfrsLVOLUME.ext
xfrdNETIF.ext
xfrsNETIF.ext
xfrdCPU.ext
xfrsCPU.ext
xfrdFILESYSTEM.ext
xfrsFILESYSTEM.ext

```
xfrdTRANSACTION.ext  
xfrsTRANSACTION.ext  
xfrdCONFIGURATION.ext
```

where **ext** = **asc** (ASCII), **dat** (datafile), **bin** (binary), or **wk1** (spreadsheet).

A special file name, **stdout** (or *), can be used with the export operation to direct the output to the **stdout** file (normally your terminal or workstation, although this can be redirected using shell commands).

```
output stdout
```

or

```
output *
```

To return the output to its default settings, type:

```
output default
```

or

```
output -
```

Note: You can override the default output file names for exported files using the output parameter in the export template file.

Note: You should not output extract operation files to **stdout**, because they are incompatible with ASCII devices. You should also not output binary or WK1 formats of the export operation to the **stdout** file for the same reason.

Note: Care should be taken to avoid appending *extracted* data to an existing *exported* data file and to avoid appending exported data to an existing extracted file. Attempts to append the wrong data type will result in an error condition.

Example

In this example, no output file is specified so the default output file, **rxlog** is used for the application summary data being extracted. The `,purge` option specifies that any existing output file should be purged.

```
extract>  
logfile /var/opt/perf/datafiles/logglob  
output rxlog,purge  
global off  
application detail  
extract month 9905
```

To perform the above task using command line arguments, enter:

```
extract -f rxlog,purge -a -xm 9905
```

process

Use the process command to specify whether or not to extract or export process data.

The default is process off.

Syntax

	[on]	
process	[detail]	[application=# [-#] ,...]
	[off]	
	[killed]	

Parameters

on	Specifies that process data <i>should</i> be extracted or exported.
detail	Specifying process detail is the same as specifying process on.
off	Specifies that process data <i>should not</i> be extracted or exported.
killed	Specifies only processes that have an interest reason that includes killed. (Processes that terminated in the measurement interval.)
application	Specifies only processes that belong to selected applications. An application can be entered as a single number or as a range of application numbers (7-9 means applications 7, 8, and 9). The application number is determined by the order of the application definition in the parm file when the data was collected. If you are specifying multiple applications, separate each one with a comma.

Note: Process data can increase the size of an extracted log file significantly. If you plan to copy the log file to a workstation for analysis, you might want to limit the amount of process data extracted.

Example

In this example, the process command specifies processes that terminated during an interval and belong to applications 1, 4, 6, 7, 8, or 10. Use the **utility** program's scan command to find the application numbers for specific applications.

```
process killed applications=1,4,6-8,10
```

quit

Use the quit command to terminate the **extract** program. The quit command is equivalent to the **extract** program's exit command.

Syntax

```
quit
```

```
q
```

report

Use the report command to specify the export template file to be used by the export function. If no export template file is specified, the default export template file, **reptfile**, is used. The export template file is used to specify various output format attributes used in the export function. It also specifies which metrics will be exported.

If you are in interactive mode and specify no export template file, all metrics for the data types requested will be exported in ASCII format.

Syntax

```
report [exporttemplatefile] [,show]
```

Parameters

,show	Specifies that the field positions and starting columns should be listed for all metrics specified in the export template file. This listing can be used when export files are processed by other programs.
-------	---

How to Use It

When you issue this command, you are prompted by a message that asks whether or not you want to validate metrics in the export template with the previously specified log file. Validation ensures that the metrics specified in the export template file exist in the log file. This allows you to check for possible errors in the export template file. If no validation is performed, this action is deferred until you perform an export.

Note: The ,show parameter of the report command discussed here is different from the show command discussed later.

sh

Use sh to enter a shell command without exiting **extract** by typing **sh** or an exclamation point (!) followed by a UNIX shell command.

Syntax

sh or ! [shell command]

Parameters

sh ls	Executes the ls command and returns to extract . The shell command is any system command.
!ls	Same as above.
!ksh	Starts a Korn shell. Does not return immediately to extract . Type exit or CTRL-d Return to return to the extract program.

How to Use It

Following the execution of the single command, you automatically return to **extract**. If you want to issue multiple shell commands without returning to **extract** after each one, you can start a new shell.

If you issue the sh command without the name of the shell command, you are prompted to supply it. For example,

```
sh
```

```
enter SYSTEM command: ls
```

shift

Use the shift command to limit data extraction to certain hours of the day corresponding to work shifts and to exclude weekends (Saturday and Sunday).

The default is shift all day to extract data for all day, every day including weekends.

Syntax

shift	<i>[starttime-stoptime]</i> [all day] [noweekends]
--------------	--

Parameters

The *starttime* and *stoptime* parameters are entered in the same format as the time in the start command. Shifts that span midnight are permitted. If *starttime* is scheduled *after* the *stoptime*, the shift will start at the start time and proceed past midnight, ending at the *stoptime* of the next day.

all day	Specifies the default shift of 12:00 am - 12:00 am (or 00:00 -00:00 on a 24-hour clock).
noweekends	Specifies the exclusion of data which was logged on Saturdays and Sundays. If <i>noweekends</i> is entered in conjunction with a shift that spans midnight, the weekend will consist of those shifts that <i>start</i> on Saturday or Sunday.

Example

In this example, disk detail data collected between 10:00 am and 4:00 pm every day starting June 15, 1999 is extracted.

```
extract>  
logfile /var/opt/perf/datafiles/logglob  
global off  
disk detail  
shift 10:00 am - 4:00 PM  
start 6/15/99  
extract
```

To perform the above task using command line arguments, enter:

```
extract -d -b 6/15/99 -s 10:00 AM-4:00 PM -xt
```

show

Use the show command to list the names of the opened files and the status of the **extract** parameters that can be set.

Syntax

```
show [all]
```

Note: The show command discussed here is different from the ,show parameter of the report command discussed earlier.

Examples

Use show by itself to produce a list that may look like this:

```
Logfile: /var/opt/perf/datafiles/logglob  
Output: Default  
Report: Default  
List: "stdout"
```

The default starting date & time = 10/08/99 12:00 AM (LAST -30)

The default stopping date & time = 11/20/99 11:59 PM (LAST -0)

The default shift = 12:00 AM - 12:00 PM

```
GLOBAL          DETAIL          records will be processed  
APPLICATION. . . . . NO records will be processed  
PROCESS . . . . . NO records will be processed  
DISK DEVICE. . . . . NO records will be processed  
LVOLUME. . . . . NO records will be processed
```

TRANSACTION. NO records will be processed
NETIFNO records will be processed
CPUNO records will be processed
FILESYSTEM.NO records will be processed
ConfigurationNO records will be processed

Use show all to produce a more detailed list that may look like this:

Logfile: /var/opt/perf/datafiles/logglob
Global file: /var/opt/perf/datafiles/logglob,version D
Application file: /var/opt/perf/datafiles/logappl
Process file: /var/opt/perf/datafiles/logproc
Device file: /var/opt/perf/datafiles/logdev
Transaction file: /var/opt/perf/datafiles/logdev
Index file: /var/opt/perf/datafiles/logindx

System ID: homer
System Type 9000/715/ S/N 2223334442 O/S HP-UX B.10.20 A
Data collector: SCOPE/UX C.02.30
File Created: 10/08/99
Data Covers: 44 days to 11/20/99
Shift is: All Day

Data records available are:
Global Application Process Disk Volume Transaction

Maximum file sizes:
Global=10.0 Application=10.0 Process=20.0 Device=10.0
Transaction=10.0 MB

Output: Default
Report: Default
List: "stdout"

The default starting date & time = 10/08/99 11:50 AM (LAST -30)
The default stopping date & time = 11/20/99 11:59 PM(LAST - 0)
The default shift = 12:00 AM - 12:00 PM

GLOBAL.....DETAIL.....records will be processed
APPLICATION.....NO records will be processed
PROCESS.....NO records will be processed

```

DISK DEVICE.....NO records will be processed
LVOLUME.....NO records will be processed
TRANSACTION.....NO records will be processed
NETIF.....NO records will be exported
CPU.....NO records will be processed
FILESYSTEM.....NO records will be processed
Configuration .....NO records will be exported

```

Export Report Specifications:

```

Interval = 3600, Separator = " "
Missing data will not be displayed
Headings will be displayed
Date/time will be formatted
Days to exclude: None

```

start

Use the start command to set a starting date and time for the extract and export functions. The default starting date is the date 30 full days before the last date in the log file, or if less than 30 days are present, the date of the earliest record in the log file.

Syntax

start	[date [time]] [today [-day]][time]] [last [-days]][time]] [first [+days]][time]]
--------------	---

Parameters

date	The date format depends on the native language that is configured for your system. If you do not use native languages or you have set C as the default language, the data format is <i>mm/dd/yy</i> (month/day/year) such as 09/30/99 for September 30, 1999, for the extract or export function.
time	The time format also depends on the native language used. For the C language, the format is <i>hh:mm am</i> or <i>hh:mm pm</i> (hour:minute in a 12-hour format with the am or pm suffix). For example, 07:00 am is 7 o'clock in the morning. Twenty-four hour time is accepted in all languages. For example, 23:30 would be accepted for 11:30 pm.

	<p>If the format of the date or time is unacceptable, you are prompted with an example in the correct format.</p> <p>If no start time is given, midnight (12:00 am) is assumed. A starting time of midnight for a given day starts at the <i>beginning</i> of that day (00:00 on a 24-hour clock).</p>
today	Specifies the current day. The qualification of the parameter, such as today-days, specifies the number of days <i>prior</i> to today's date. For example, today-1 indicates yesterday's date and today-2, the day before yesterday.
last	Can be used to represent the last date contained in the log file. The parameter last-days specifies the number of days <i>prior</i> to the last date in the log file.
first	Can be used to represent the first date contained in the log file. The parameter first+days specifies the number of days <i>after</i> the first date in the log file.

How to Use It

The following commands override the starting date set by the start command.

- weekly
- monthly
- yearly
- extract (If day, week, month, or year parameter is used)
- export (If day, week, month, or year parameter is used)

Example

In this example, the start command specifies June 5, 1999 8:00 am as the start time of the first interval to be extracted. The output command specifies an output file named **myout**.

```
logfile /var/opt/perf/datafiles/logglob
start 6/5/99 8:00 am
output myout
global detail
extract
```

To perform the above task using command line arguments, enter:

```
extract -g -b 06/05/99 8:00 AM -f myout -xt
```

stop

Use the stop command to terminate an extract or export function at a specified date and time.

The default stopping date and time is the *last* date and time recorded in the log file.

Syntax

start	[date [time]]
--------------	----------------------

	<p>[today [-day][time]]</p> <p>[last [-days][time]]</p> <p>[first [+days][time]]</p>
--	--

Parameters

date	<p>The date format depends on the native language that is configured for your system. If you do not use native languages or you have set C as the default language, the data format is <i>mm/dd/yy</i> (month/day/year) such as 09/30/99 for September 30, 1999, for the extract or export function.</p>
time	<p>The time format also depends on the native language used. For the C language, the format is <i>hh:mm am</i> or <i>hh:mm pm</i> (hour:minute in a 12-hour format with the am or pm suffix). For example, 07:00 am is 7 o'clock in the morning.</p> <p>Twenty-four hour time is accepted in all languages. For example, 23:30 would be accepted for 11:30 pm.</p> <p>If the format of the date of time is unacceptable, you are prompted with an example in the correct format.</p> <p>If no stop time is given, one minute before midnight (11:59 pm) is assumed. A stopping time of midnight (12:00 am) for a given day stops at the <i>end</i> of that day (23:59 on a 24-hour clock).</p>
today	<p>Specifies the current day. The qualification of the parameter, such as today-days, specifies the number of days prior to today's date. For example, today-1 indicates yesterday's date and today-2 the day before yesterday.</p>
last	<p>Can be used to represent the last date contained in the log file. The parameter last-days specifies the number of days <i>prior</i> to the last date in the log file.</p>
first	<p>Can be used to represent the first date contained in the log file. The parameter first+days specifies the number of days <i>after</i> the first date in the log file.</p>

How to Use It

The following commands override the stopping date set by the *stop* command.

- weekly
- monthly
- yearly
- extract (If day, week, month, or year parameter is used)
- export (If day, week, month, or year parameter is used)

Example

In this example, the *stop* command specifies June 5, 1999 5:00 pm as the stopping time of the last interval to be extracted. The *output* command specifies an output file named **myout**.

```
extract>
```

```
logfile /var/opt/perf/datafiles/logglob
```

```
start 6/5/99 8:00 AM
```

```
stop 6/5/99 5:00 PM
```

```
output myout
```

```
global detail
```

```
extract
```

To perform the above task using command line arguments, enter:

```
extract -g -b 6/5/99 8:00 AM -e 6/5/99 5:00 PM -f myout -xt
```

transaction

Use the transaction command to specify the type of transaction data that is being extracted or exported.

Syntax

transaction	[on] [detail] [summary] [both] [off]
--------------------	--

Parameters

on or detail summary both off	See "Parameters" in the description of the "application" on page 161 command at the beginning of this chapter. Summary and both can only be exported.
--	---

Example

A new extracted log file called **rxmay99** is created on June 1, 1999. Any existing file that has this name is purged. All raw transaction log file data collected from May 1, 1999 to May 31, 1999 is extracted.

```
extract>
```

```
logfile /var/opt/perf/datafiles/logglob
```

```
output rxmay99,purge
```

```
global detail
```

```
transaction detail
```

```
month 9905
```

To perform the above task using command line arguments, enter:

```
extract -gt -f rxmay99,purge -xm 9905
```

weekdays

Use the weekdays command to exclude data for specific days from being exported (day 1 = Sunday).

Syntax

```
weekdays [1|2.....7]
```

How to Use It

If you want to export data from only certain days of the week, use this command to exclude the days from which you *do not* want data. Days have the following values:

Sunday =1

Monday =2

Tuesday =3

Wednesday =4

Thursday =5

Friday =6

Saturday =7

For example, if you want to export data that was logged only on Monday through Thursday, *exclude* data from Friday, Saturday, and Sunday from your export.

Example

In this example, any detailed global data logged on Tuesdays and Thursdays is excluded from the export. The output export file contains the global metrics specified in the **myrept** export template file.

```
extract>
```

```
logfile /var/opt/perf/datafiles/logglob
```

```
global detail
```

```
report myrept
```

```
weekdays 35
```

```
export
```


weekly

Use the `weekly` command to specify data extraction based on a calendar week. A week is defined as seven days starting on Monday and ending on Sunday.

During execution, this command sets the start and stop dates to the proper dates, based on the week and year of the extracted data.

Syntax

<code>weekly</code>	<code>[yyww]</code> <code>[ww]</code>
---------------------	--

Parameters

<code>weekly</code>	Extracts the current week's data (the default).
<code>weeklyww</code>	Extracts data for a specific week from this year's data (where <code>ww</code> is any number from 01 to 53).
<code>weeklyyyww</code>	Extracts data for a specific week <i>and</i> year (where <code>yyww</code> is a single number consisting of the last two digits of the year and the two-digit week-of-the-year number). For example, the 20th week of 1999 would be <code>weekly 9920</code> .

If you do not specify the log file before executing the `weekly` command, the default **logglob** file in the **datafiles** directory is used.

How to Use It

Use the `weekly` command when you are extracting data for archiving on a weekly basis.

The name of the output file consists of the letters **rxwe** followed by the last two digits of the year, and the two-digit week number for the week being extracted. For example, the 12th week of 1999 (from Monday, March 22 to Sunday, March 29) would be output to a file named **rxwe9912**.

The type of data extracted and summarized follow the normal rules for the `extract` command and can be set before executing the `weekly` command. These settings are honored unless a weekly output file already exists. If it does, data is appended to it, based on the type of data selected originally.

The `weekly` command has a feature that opens the *previous* week's extracted file and checks to see if it is filled--whether it contains data extracted up to the last day of the week. If not, the `weekly` command appends data to this file to complete the previous week's extraction.

For example, a `weekly` command is executed on Thursday, May 20, 1999. This creates a log file named **rxwe199920** containing data from Monday, May 17 through the current date (May 20).

On Wednesday, May 26, 1999, another `weekly` command is executed. Before the **rxwe199921** file is created for the current week, the **rxwe199920** file from the previous week is opened and checked. When it is found to be incomplete, data is appended to it to complete the extraction through Sunday, May 23, 1999. Then, the **rxwe199921** file is created to hold data from Monday, May 24, 1999 to the current date (May 26).

As long as you execute the weekly command at least once a week, this feature will complete each week's file before creating the next week's file. When you see two adjacent weekly files (for example, **rxwe199920** and **rxwe199921**), you can assume that the first file is complete for that week, and it can be archived and purged.

Note: The weeks are numbered based on their starting day. Thus, the first week of the year (week 01) is the week starting on the *first* Monday of that year. Any days before that Monday belong to the last week of the previous year. The weekly and extract week commands are similar in that they both extract one calendar week's data. The weekly command ignores the setting of the output command, using instead predefined output file names. It also attempts to append missing data to the previous week's extracted log file if it is still present on the system. The extract week command, on the other hand, uses the settings of the output command. It cannot append data to the previous week's extracted file because it does not know its name. The output file is named **rxwe** followed by the current year (*yyyy*) and week of the year (*ww*).

Example

In this example, the weekly command causes the current week's data to be extracted and complete the previous week's extracted file, if it is present.

```
extract>  
logfile /var/opt/perf/datafiles/logglob  
global detail  
application detail  
process detail  
weekly
```

To perform the above task using command line arguments, enter:

```
extract -gap -xw
```

yearly

Use the yearly command to specify data extraction based on a calendar year.

During execution, the command sets the start and stop dates to the proper dates, based on the year being extracted.

Syntax

yearly	[yyyy] [yy]
---------------	------------------------------

Parameters

yearly	Extracts the current year's data (the default).
yearly	Extracts a specific year's data (where yy is a number from 00 to 99). The specifications 00

<code>yy</code>	to 27 assume the years 2000 to 2027, whereas 71 to 99 assume the years 1971 to 1999.
<code>yearly</code> <code>yyyy</code>	Extracts a specific year's data (where <code>yyyy</code> is the full-year numbered 1971 to 2027).

If you do not specify the log file before executing the `yearly` command, the default **logglob** file is used.

How to Use It

Use the `yearly` command when you are extracting data for archiving on a yearly basis.

The name of the output file consists of the letters **rxyr** followed by the four digits of the year being extracted. Thus, data from 1999 would be output to a file named **rxyr1999**.

The type of data extracted and summarized follow the normal rules for the `extract` command and can be set before executing the `yearly` command. These settings are honored unless a yearly output file already exists. If it does, data is appended to it, based upon the type of data selected originally.

The `yearly` command has a feature that opens the *previous* year's extracted file and checks to see if it is filled—whether it contains data extracted up to the last day of the year. If not, the `yearly` command appends data to this file to complete the previous year's extraction.

For example, a `yearly` command was executed on December 15, 1998. This created a log file named **rxyr1998** containing data from January 1, 1998 to the current date (December 15).

On January 5, 1999, another `yearly` command is executed. Before the **rxyr1999** file is created for the current year, the **rxyr1998** file from the previous year is opened and checked. When it is found to be incomplete, data is appended to it to complete its extraction until December 31, 1998. Then, the **rxyr1999** file is created to hold data from January 1, 1999 to the current date (January 5).

As long as you execute the `yearly` command at least once a year, this feature will complete each year's file before creating the next year's file. When you see two adjacent yearly files (for example, **rxyr1998** and **rxyr1999**), you can assume that the first file is complete for that year, and it can be archived and purged.

The previous paragraph is true *only* if the raw log files are sized large enough to hold *one full year* of data. It would be more common to size the raw log files smaller and execute the `yearly` command more often (such as once a month).

Note: The `yearly` and `extract year` commands are similar in that they both extract one calendar year's data. The `yearly` command ignores the setting of the output command, using instead predefined output file names. It also attempts to append missing data to the previous year's extracted log file if it is still present on the system. The `extract year` command, on the other hand, will use the settings of the output command. It cannot append data to the previous year's extracted file since it does not know its name.

Example

In this example, application and global detail data is appended to the existing yearly summary file (or creates it, if necessary). The output file is **rxyr`yyyy`** (where `yyyy` represents the current year).

```
extract>  
logfile /var/opt/perf/datafiles/logglob  
global detail
```

application detail

process off

yearly

To perform the above task using command line arguments, enter:

extract -ga -xy

Chapter 9: Using the cpsh Program

You can use the cpsh program only if you enable the HP Ops OS Inst to Realtime Inst LTU or Glance Pak Software LTU.

The cpsh program presents a new command-line prompt, which enables you to view the real-time metric data collected from the monitored system.

Using the Interactive Mode

You can use the cpsh program in the interactive mode. If you run cpsh command without any options, the cpsh program opens a new command prompt. At this prompt, you can perform different tasks to view the details of the real-time metrics.

To open the cpsh prompt, follow these steps:

1. Log on to a system (with the root/administrative privileges) where the HP Operations agent is installed.
2. Run the following command to open the cpsh prompt of the local system:

cpsh

Run the following command to open the cpsh prompt of a remote system:

cpsh -n <system_name>

where <system_name> is the fully-qualified domain name of the remote system.

or

cpsh -n <ip_address>

where <ip_address> is the IP address of the remote system.

Note: While opening the cpsh prompt of a remote system, make sure that the perfd process runs on the remote system. You can prevent other systems from accessing the performance data of the local system through the cpsh utility. For more information, see ["Restrict Access" on page 82](#).

The cpsh prompt opens.

To view metrics data in a well-structured format, run the **cpsh** command with the -t option.

For example:

cpsh -t

or

cpsh -n <system_name> -t

3. To view the details of the available commands for use with the cpsh prompt, type **help**.

View Real-Time Metrics

You can view the real-time values of the available metrics from the csh prompt. Before you perform any operation at the csh prompt, you must set the metric context. The perfd daemon and associated utilities process the available data based on metric classes. Therefore, while using the csh utility to view real-time data, you must always set the metric class before performing any operations to view the available data.

To view the real-time values of the metrics of a metric class, follow these steps:

1. At the csh prompt, type **class** *<metric_class>*.
2. List of currently set metrics for the given class, type **list**. The list of all default metrics for the specified metric class appears.
3. To view values of the metrics that belong to the specified class, type **push** at the csh prompt. The csh program displays real-time values of the metrics in a tabular format.
4. To go back to the csh prompt, press **Ctrl+C**.

Modify a Metric Class

You can add additional available metrics to the list of default metrics for a metric class. To add or delete a metrics from a metric class at the csh prompt, follow these steps:

1. Open the csh prompt.
2. At the csh prompt, type **class** *<metric_class>*.
3. Type **list**. The list of all default metrics for the specified metric class appears.
4. To delete a metric, follow these steps:
At the csh prompt, type **delete** *<metric_name>*.
Type **list**. The list of metrics for the specified metric class does not include the deleted metric.
5. To add a metric to the metric class, follow these steps:
At the csh prompt, type **add** *<metric_name>*.
Type **list**. The list of metrics for the specified metric class includes the newly added metric.

View All the Available Metrics

To view all the available metrics that belong to a metric class, follow these steps:

1. Open the csh prompt.
2. At the csh prompt, type **class** *<metric_class>*.
3. Type **list all**. The list of all available metrics that belong to the specified metric class appears.

Organize a Metric Class

You can reorganize a metric class without performing sequential add and delete operation on the class. To reorganize a metric class to include the metrics of your choice, follow these steps:

1. Open the csh prompt.
2. At the csh prompt, type **class** *<metric_class>*.
3. Type **init** *<metric_name>* *<metric_name>* *<metric_name>*

The specified metric class incorporates only the metrics specified with the init command.

View Metric Help

You can view the description of every real-time metric from the csh prompt. To view the metric description, follow these steps:

1. Open the csh prompt.
2. Type **class** *<metric_class>*.
3. Type **help** *<metric_name>* at the csh prompt. The details description of the metric appears.

View Summarized Metric Data

For the metrics of the GLOBAL and TABLE classes, you can view summarized data from the csh prompt. To view the summarized data, follow these steps:

1. Open the csh prompt.
2. At the csh prompt, type **class gbl** or **class tbl**.
3. Type **summ** *<interval>*. In this instance, *<interval>* is the summarization interval specified in seconds. *<interval>* must be a multiple of collection interval of perfd server to which csh is connected.

The csh utility displays the following measurements of the values of metrics that belong to the selected metric class:

Maximum

Minimum

Average

Standard deviation

Enabling Threshold and Filters Conditions

You can set the threshold and filter options based on your requirement in the Perfd process and view the data available with the set qualifying criteria.

Set the following options:

Threshold - This option is only available for metrics of the GLOBAL class. You can specify the global threshold and view the data available above the set global threshold value.

Filter - The filter option is available for metrics of all the classes.

You can set the conditions as:

- **filter** <metric> <operator> <value>
- **threshold** <metric> <operator> <value>

You can also set both the conditions to view only data that meets the set criteria.

Example 1: List the number of processes where utilization is more that 4 percent.

Use the filter condition to view data.

To view the filtered data, follow these steps:

1. Open the csh prompt.
2. At the csh prompt, type **class proc**
3. Type **filter proc_cpu_total_util > 4**

The csh utility displays the processes which have CPU utilization more than 4 percent.

4. Type **push**

The results appear as:

User	App	Interest	Process	ProcessName
Name	PID	ID Reason	Start Time	CPU Name
root	1028	4	05/07/2013 07:17:35	5.2 find

Example 2: List the number of processes where utilization is more that 4 percent and threshold is set to 1 percent.

Set the threshold option and then the filter option to view data.

To view the filtered data, follow these steps:

1. Open the csh prompt.
2. At the csh prompt, type **class proc**
3. Type **threshold proc_cpu_total_util > 1**
4. Type **filter proc_cpu_total_util > 3**

The csh utility displays the processes where the threshold value is set to more than 1 percent and also the CPU utilization is more than 3 percent.

5. Type **push**

The results appear as:

User	App	Interest	Process	ProcessName
Name	PID	ID Reason	Start Time	CPU Name
root	1028	4	05/07/2013 07:17:35	4.9 find

Chapter 10: Adviser for the RTMA Component

You can use the adviser feature only if you enable the HP Ops OS Inst to Realtime Inst LTU or Glance Pak Software LTU.

The adviser feature enables you to generate and view alarms when values of certain metrics, collected by the RTMA component, exceed (or fall below) the set threshold. The **adviser script** and **padv** utility build up the adviser feature. The adviser script helps you create the rules to generate alarms when the performance of the monitored system shows signs of degradation. The **padv** utility helps you run the adviser script on a system of your interest.

Note: This topic focuses on the adviser feature that can be used with the RTMA component. The GlancePlus software offers additional features that can be used with the adviser utility. For information on using the adviser feature with the GlancePlus software, see *online help for GlancePlus*.

Alarms and Symptoms

Alarms enable you to highlight metric conditions. The adviser script enables you to define threshold values for metrics that are monitored by the RTMA component. When the metric value traverses beyond the set threshold, the RTMA component generates an alarm in the form of an alert message. This message is sent in the form of `stdout` to the **padv** utility.

An alarm can be triggered whenever conditions that you specify are met. Alarms are based on any period of time you specify, which can be one interval or longer.

A symptom is a combination of conditions that affects the performance your system.

By observing different metrics with corresponding thresholds and adding values to the probability that these metrics contribute to a bottleneck, the adviser calculates one value that represents the combined probability that a bottleneck is present.

Working of the Adviser Script

When you run the **padv** command, the HP Operations agent scans the script specified with the command and takes necessary actions. If you do not specify any script file with the **padv** command, the adviser utility retrieves necessary information from the following default script file:

- *On Windows:* `%ovdatadir%\perf\perfd`
- *On UNIX and Linux:* `/var/opt/perf/perfd`

If you want to run the script that includes operating system-specific diagnosis and actions, use the following default scripts:

- *On Windows:* %ovdatadir%\perf\perfd\os\\adv
- *On UNIX and Linux:* /var/opt/perf/perfd/os//adv

In this instance, <os_type> is the type of operating system of node where you want to run the script.

As a result of running the adviser script, you can achieve the following:

- Print the system status based on generated alarms into a text file
- View the real-time status of the system in the command console from where you ran the padv command.

Using Adviser

To use the adviser component to monitor the real-time health of the system, follow these steps:

1. Configure the adviser script according to your requirements. Sample scripts are available into the following directory:

- *On Windows:* %ovinstalldir%\examples\adviser
- *On UNIX or Linux:* /opt/perf/examples/adviser

2. Identify the node where you want to run the script.
3. Make sure the perfd process runs on the identified system.
4. Run the following command:

```
padv -s <script_name>-n<system_name>
```

The adviser script starts running on the specified system and creates results based on the configuration of the script file.

Tip: While using the scripts on a remote system, make sure that the perfd process runs on the remote system. You can prevent other systems from running the adviser script on the local system. For more information, see [Restrict Access](#).

Run the Adviser Script on Multiple Systems

You can use the mpadv command to run the adviser script on multiple systems. To use the mpadv command, follow these steps:

1. Identify the nodes where you want to run the script.
2. Create a text file listing the names of all the identified systems.
3. Save the text file on the local system.
4. Configure the adviser script according to your requirements. Sample scripts are available into the following directory:
 - *On Windows:* %ovinstalldir%\examples\adviser
 - *On UNIX or Linux:* /opt/perf/examples/adviser
5. Make sure the perfd process runs on the identified system.

6. Run the following command:

```
mpadv -l <system_list_text_file> -s <script_name>
```

The adviser script starts running on the systems specified in the *<system_list_text_file>* file and creates results based on the configuration of the script file.

Adviser Syntax

The adviser syntax is a simple script language that enables you to set alarms and define symptom conditions.

A default syntax file— *adviser.syntax*— is provided in the following directory:

- *On Windows:* %ovdatadir%\perf
- *On UNIX and Linux:* /var/opt/perf

You can edit the syntax file to define your own alarms and symptoms.

Syntax Conventions

- Braces ({}) indicate that one of the choices is required.
- Brackets ([]) indicate an optional item.
- Items separated by commas within brackets or braces are options. Choose only one.
- Italics indicate a variable name that you will replace.
- Adviser syntax keywords must always be written in the capital case.

Comments

Syntax:

```
# [any text or characters]
```

or

```
// [any text or characters]
```

You can precede comments either by double forward slashes (//) or the # sign. In both cases, the comment ends at the end of the line.

Conditions

A condition is defined as a comparison between two metric names, user variables, or numeric constants.

```
item1 {>, <, >=, <=, ==, !=} item2 [OR item3 \  
{>, <, >=, <=, ==, !=} item4]
```

or:

```
item1 {>, <, >=, <=, ==, !=} item2 [AND item3 \  
    {>, <, >=, <=, ==, !=} item4]  
(==" means "equal", and "!=" means "not equal".)
```

Conditions are used in the ALARM statement and the IF statement. They can be used to compare two numeric metrics, variables or constants, and they can also be used between two string metric names, user variables or string constants. For string conditions, only == or != can be used as operators.

You can use compound conditions by specifying the OR or AND operator between subconditions.

Examples:

```
gbl_swap_space_reserved_util > 95  
proc_proc_name == "test" OR proc_user_name == "tester"  
proc_proc_name != "test" AND  
    proc_cpu_sys_mode_util > highest_proc_so_far
```

Constants

Constants can be either alphanumeric or numeric. An alphanumeric constant must be enclosed in double quotes. There are two kinds of numeric constants: integer and real. Integer constants may contain only digits and an optional sign indicator. Real constants may also include a decimal point.

Examples:

345	Numeric integer
345.2	Numeric real
"Time is"	Alphanumeric literal

Expressions

Use expressions to evaluate numerical values. An expression can be used in a condition or an action.

An expression can contain:

- Numeric constants
- Numeric metric names
- Numeric variables
- An arithmetic combination of the above
- A combination of the above grouped together using parentheses

Examples:

```
Iteration + 1  
3.1416  
gbl_cpu_total_util - gbl_cpu_user_mode_util
```

```
( 100 - gbl_cpu_total_util ) / 100.0
```

Metric Names in Adviser Syntax

You can directly reference metrics anywhere in the Adviser syntax. You can use the following types of metrics in the Adviser syntax:

- Global metrics (prefixed with gbl_ or tbl_)
- Application metrics (prefixed with app_)
- Process metrics (prefixed with proc_)
- Disk metrics (prefixed with bydisk_)
- By CPU metrics (prefixed with bycpu_)
- File system metrics (prefixed with fs_)
- Logical volume metrics (prefixed with lv_)
- Network interface metrics (prefixed with bynetif_)
- Swap metrics (prefixed with byswp_)
- ARM metrics (prefixed with tt_ or ttbin_)
- PRM metrics (prefixed with prm_)
- Locality Domain metrics (prefixed by ldom_)

You can only use process, logical volume, disk, file system, LAN, and swap metrics within the context of a LOOP statement.

Metric names can contain alphanumeric (for example, gbl_machine or app_name) or numeric data and can reflect several different kinds of measurement. For example, the metric ending of a metric name indicates what is being measured:

- a `_util` metric measures utilization in percentages
- a `_rate` metric measures units per second
- a `_queue` metric measures the number of processes or threads waiting for a resource

If you are unsure of the unit of measure for a specific metric, refer to the metric definition document.

You must associate an application metric with a specific application, except when using the LOOP statement. To do this, specify the application name followed by a colon, and then the metric name. For example, `other_apps:app_cpu_total_util` specifies the total CPU utilization for the application `other_apps`. Refer to the ALIAS statement description for more information on using application metrics in the syntax.

Application names, as defined by the parm file, may contain special characters and embedded blanks. To use these names in the syntax (where application names must match the form of a variable name), the names are made case-insensitive and embedded blanks are converted to underlines. This means that the application name defined as "Other Apps" may be referenced in the syntax as `other_apps`. For application names defined with special characters, you must use the ALIAS statement to specify an alternate name.

When explicitly qualified, application metrics may be referenced anywhere in the syntax. Unqualified application metrics may only be referenced within the context of the LOOP statement. This is an iterative statement which implicitly qualifies application or process metrics.

You can only reference process metrics within the context of a LOOP statement. There is no way to explicitly reference a process.

Printlist

The printlist is any combination of properly formatted expressions, Metric Names, user variables, or constants. See the examples for the proper formatting.

- Expression examples:

```
expression [|width[|decimals]]
```

Metric Names or User Variable examples:

```
metric names [|width[|decimals]]
```

or

```
user variables [|width[|decimals]]
```

The metric names or user variables must be alphanumeric.

- Constant examples:

No formatting is necessary for constants.

Formatted Examples:

```
gbl_cpu_total_util|6|2    formats as  '100.00'
```

```
(100.32 + 20)|6    formats as      ' 120'
```

```
gbl_machine|5    formats as          '7013/'
```

```
"User Label"    formats as      "User Label"
```

Variables

Variables must begin with a letter and can include letters, digits, and the underscore character.

Variables are not case-sensitive.

Define a variable by assigning something to it. The following example defines the numeric variable `highest_CPU_value` by assigning it a value of zero.

```
highest_CPU_value = 0
```

The following example defines the alphanumeric variable `my_name` by assigning it a null string value.

```
my_name = ""
```

ALARM Statement

Use the ALARM statement to notify you when certain events, which you define, occur on your system.

Using the ALARM statement, the adviser script can notify you through messages sent to the originating console of the `padv` command.

Syntax:

```
ALARM condition [FOR duration {SECONDS, MINUTES, INTERVALS}]  
  [condition [FOR duration {SECONDS, MINUTES, INTERVALS}] ] ...
```

```
[START statement]
```

```
[REPEAT [EVERY duration [SECONDS, MINUTES, INTERVAL, INTERVALS]]  
  statement]
```

```
[END statement]
```

The ALARM statement must be a top-level statement. It cannot be nested within any other statement.

However, you can include several ALARM conditions in a single ALARM statement, in which case all conditions must be true for the alarm to trigger. And you can also use a COMPOUND Statement, which is executed at the appropriate time during the alarm cycle.

START, REPEAT, and END are ALARM statement keywords. Each of these keywords specifies a statement. You must have a START, REPEAT, or END in an ALARM statement, and they must be listed in correct order.

The alarm cycle begins on the first interval that all of the alarm conditions have been true for at least the specified duration. At that time, the adviser script executes the START statement, and on each subsequent interval checks the REPEAT condition. If enough time has transpired, the statement for the REPEAT clause is executed. This continues until one or more of the alarm conditions becomes false. This completes the alarm cycle and the END statement is executed.

If you omit the EVERY specification from the REPEAT statement, the adviser script executes the REPEAT statement at each interval.

ALERT Statement

The ALERT statement is used to place a message in padv command console. Whenever an ALARM detects a problem, it can run an ALERT statement to send a message with the specified severity to the padv command console.

You can use the ALERT statement in conjunction with an ALARM statement.

Syntax:

```
[(RED or CRITICAL), (YELLOW or WARNING), RESET] ALERT printlist  
RED and YELLOW, are synonymous with CRITICAL and WARNING.
```

ALIAS Statement

Use the ALIAS statement to assign a variable to an application name that contains special characters or imbedded blanks.

Syntax:

```
ALIAS variable = "alias name"
```

ALIAS Example

Because you cannot use special characters or embedded blanks in the syntax, using the application name "other user root" in the PRINT statement below would have caused an error. Using ALIAS, you can still use "other user root" or other strings with blanks and special characters within the syntax.

```
ALIAS otherapp = "other user root"
```

```
PRINT "CPU for other root login processes is: ",  
      otherapp:app_cpu_total_util
```

ASSIGNMENT Statement

Use the ASSIGNMENT statement to assign a numeric or alphanumeric value, expression, to the user variable.

Syntax:

```
[VAR] variable = expression
```

```
[VAR] variable = alphaitem
```

```
[VAR] variable = alphaitem
```

COMPOUND Statement

Use the COMPOUND statement with the IF statement, the LOOP statement, and the START, REPEAT, and END clauses of the ALARM statement. By using a COMPOUND statement, a list of statements can be executed.

Syntax

```
{  
statement  
statement  
}
```

Construct compound statements by grouping a list of statements inside braces ({}). The compound statement can then be treated as a single statement within the syntax.

Compound statements cannot include ALARM and SYMPTOM statements. (Compound is a type of statement and not a keyword.)

EXEC Statement

Use the EXEC statement to execute a UNIX command from within your Adviser syntax. You could use the EXEC command, for example, if you wanted to send a mail message to the MIS staff each time a certain condition is met.

Syntax

```
EXEC printlist
```


The resulting printlist is submitted to your operating system for execution.

Because the EXEC command you specify may execute once every update interval, be careful when using the EXEC statement with operating system commands or scripts that have high overhead.

IF Statement

Use the IF statement to test conditions you define in the adviser script syntax.

Syntax:

```
IF condition THEN statement [ELSE statement]
```

The IF statement tests the condition. If true, the statement after the THEN is executed. If the condition is false, then the action depends on the optional ELSE clause.

If an ELSE clause has been specified, the statement following it is executed. Otherwise, the IF statement does nothing. The statement can be a COMPOUND statement which tells the adviser script to execute multiple statements.

LOOP Statement

Use LOOP statements to find information about your system. For example, you can find the process that uses the highest percentage of CPU or the swap area that is being utilized most. You find this information with the LOOP statement and with corresponding statements that use metric names for the system conditions on which you are gathering information.

Syntax:

```
{APPLICATION, APP, CPU, DISK, DISK_DETAIL, FILESYSTEM, FS, FS_DETAIL, LAN,  
LOGICALVOLUME, LV, LV_DETAIL, NETIF, NFS, NFS_BYSYS_OPS, NFS_OP, PRM,  
PRM_BYVG, PROCESS, PROC, PROC_FILE, PROC_REGION, PROC_SYSCALL, SWAP,  
SYSTEMCALL, SC, THREAD, TRANSACTION, TT, TTBIN, TT_CLIENT, TT_INSTANCE,  
TT_UDM, TT_RESOURCE, TT_INSTANCE_CLIENT, TT_INSTANCE_UDM, TT_CLIENT_UDM,  
LDM, PROC_LDM}
```

```
LOOP statement
```

A LOOP can be nested within other syntax statements, but you can only nest up to five levels. The statement may be a COMPOUND statement which contains a block of statements to be executed on each iteration of the loop. A BREAK statement allows the escape from a LOOP statement.

If you have a LOOP statement in your syntax for collecting specific data and there is no corresponding metric data on your system, the adviser script skips that LOOP and continues to the next syntax statement or instruction. For example, if you have defined a LOGICAL VOLUME LOOP, but have no logical volumes on your system, the adviser script skips that LOGICAL VOLUME LOOP and continues to the next syntax statement.

Loops that do not exist on your platform generate a syntax error.

As LOOP statement iterates through each interval, the values for the metric used in the statement change. For example, the following LOOP statement executes the PRINT statement once for each active application on the system, causing the name of each application to be printed.

PRINT Statement

Use the PRINT statement to print to stdout (the padv command console) the data you are collecting. You may want to use the PRINT statement to log metrics or calculated variables.

Syntax:

```
PRINT printlist
```

PRINT Example

```
PRINT "The Application OTHER has a total CPU of ",  
      other:app_cpu_total_util, "%"
```

When started, this statement prints a message to the padv command console as follows:

```
The Application OTHER has a total CPU of 89%
```

SYMPTOM Statement

Syntax:

```
SYMPTOM variable [TYPE = {CPU, DISK, MEMORY, NETWORK}]
```

```
RULE measurement {>, <, >=, <=, ==, !=} value PROB probability
```

```
[RULE measurement {>, <, >=, <=, ==, !=} value PROB probability]
```

```
.  
. .  
. . .
```

The keywords SYMPTOM and RULE are exclusive for the SYMPTOM statement and cannot be used in other syntax statements. The SYMPTOM statement must be a top-level statement and cannot be nested within any other statement.

variable is a variable name that will be the name of this symptom. Variable names defined in the SYMPTOM statement can be used in other syntax statements, but the variable value should not be changed in those statements.

RULE is an option of the SYMPTOM statement and cannot be used independently. You can use as many *RULE* options within the SYMPTOM statement as you need.

The SYMPTOM variable is evaluated according to the RULEs at each interval.

Measurement is the name of a variable or metric that is evaluated as part of the RULE

Value is a constant, variable, or metric that is compared to the measurement

Probability is a numeric constant, variable, or metric

The probabilities for all true SYMPTOM RULEs are added together to create a SYMPTOM value. The SYMPTOM value then appears in the message in the `padv` command console.

The sum of all probabilities where the condition between measurement and value is true is the probability that the symptom is occurring.

Chapter 11: Performance Alarms

You can use the Performance Collection Component to define alarms. These alarms notify you when scope or DSI metrics meet or exceed conditions that you have defined.

To define alarms, you must specify conditions on each monitored system that, when met, trigger an alert or action. You can define alarms in the alarm definitions text file, `alarmdef`.

As data is logged by `scope` or other collectors, it is compared to the alarm definitions to determine if a condition is met. When this occurs, an alert or action is triggered.

With the real-time alarm generator, you can perform the following tasks:

- Send alert notifications to the HPOM console
- Create an SNMP trap when an alert notification is generated
- Forward the SNMP trap to an SNMP trap listener
- Perform local actions on the monitored systems

You can analyze historical log file data against the alarm definitions and report the results using the utility program's `analyze` command.

Processing Alarms

As performance data is collected by the Performance Collection Component, the collected data is compared to the alarm conditions defined in the `alarmdef` file to determine whether the conditions were met. When a condition is met, an alarm is generated and the actions defined for alarms (ALERTs, PRINTs, and EXECs) are performed.

As data is collected in the log files, it is compared to the alarm definitions in the `alarmdef` file. When an alarm condition is met, the actions defined in the alarm definition are carried out. However, if data is not logged into the log files (for instance, when the threshold parameters are set to a high value), alarms are not generated even if the alarm conditions in the `alarmdef` file are met. See [Thresholds](#) for the thresholds of different classes of metrics.

Actions defined in the alarm definition can include:

- local actions performed by using operating system commands
- messages sent to Network Node Manager (NNM) and HPOM

Alarm Generator

The Performance Collection Component alarm generator handles the communication of alarm notifications. The alarm generator consists of the alarm generator server (`perfa1arm`), the alarm generator database (`agdb`), and the utility program `agsysdb`.

The `agdb` contains a list of SNMP nodes. The `agsysdb` program is used for displaying and changing the actions taken by alarm events.

When you start up Performance Collection Component, `perfa1arm` starts and reads the `agdb` at startup to determine where and whether to send alarm notifications.

Use the following command line option to see a list showing where alert notifications are being sent:

```
agsysdb -l
```

Sending SNMP Traps to Network Node Manager

To send SNMP traps to Network Node Manager, you must add your system name to `agdb` in Performance Collection Component using the command:

```
agsysdb -add systemname
```

Every ALERT generated will cause an SNMP trap to be sent to the system you defined. The trap text will contain the same message as the ALERT.

To stop sending SNMP traps to a system, you must delete the system name from `agdb` using the command:

```
agsysdb -delete systemname
```

Sending Messages to HPOM

You can have alert notifications sent to HPOM. By default, the alarm generator does *not* execute local actions that are defined in any alarms in the EXEC statement. Instead, it sends a message to HPOM's event browser.

You can change the default to stop sending information to HPOM using the following command:

```
agsysdb -ovo OFF
```

To send Performance Collection Component traps to another node, add the following entries to `/etc/services` file.

```
snmp-trap 162/tcp # SNMPTRAP
```

```
snmp-trap 162/udp # SNMPTRAP
```

In this instance, 162 specifies port number. If you want Performance Collection Component to send traps to another node, it checks the `/etc/services` file for the `snmp-trap` string. If this entry is not available, the traps will not be sent to another node.

Executing Local Actions

By default, the Performance Collection Component does not run the local commands specified in the EXEC statements.

You can change the default to enable local actions as follows:

```
agsysdb -actions always
```

The following table lists the settings for sending information to HP Operations Manager (HPOM) and for executing local actions:

Table 9: Settings for sending information to HPOM and executing local actions

Flags	Operations Monitoring Component Running	Operations Monitoring Component Not Running
HPOM Flag		
off	No alert notifications sent to HPOM.	No alert notifications sent to HPOM.
on	Alert notifications sent to HPOM.	No alert notifications sent to HPOM.
Local Actions Flag		
off	No local actions executed.	No local actions executed.
always	Local actions executed even if the Operations Monitoring Component is running.	Local actions executed.
on	Local actions sent to HPOM.	Local actions executed.

Errors in Processing Alarms

The last error that occurred when sending an alarm is logged in agdb. To view the contents of agdb, type:

```
agsysdb -l
```

The following information is displayed:

```
PA alarming status:
```

```
OVO messages : on      Last Error : none
```

```
Exec Actions : on
```

```
Analysis system: <hostname>, Key=<ip address>
```

```
PerfView : no Last Error : <error number>
```

```
SNMP : yes Last Error : <error number>
```

Analyzing Historical Data for Alarms

You can use the utility program's `analyze` command to find alarm conditions in log file data (see [Utility Commands](#)). This is different from the processing of real-time alarms explained earlier because you are comparing historical data in the log file to the alarm definitions in the `alarmdef` file to determine what alarm conditions would have been triggered.

Examples of Alarm Information in Historical Data

The following examples show what is reported when you analyze alarm conditions in historical data.

For the first example, `START`, `END`, and `REPEAT` statements have been defined in the alarm definition. An alarm-start event is listed every time an alarm has met all of its conditions for the specified duration. When these conditions are no longer satisfied, an alarm-end event is listed. If an alarm condition is satisfied for a period long enough to generate another alarm without having first ended, a repeat event is listed.

Each event listed shows the date and time, alarm number, and the alarm event. `EXEC` actions are *not* performed, but they are listed with any requested parameter substitutions in place.

```
05/10/99 11:15 ALARM [1] START
CRITICAL: CPU test 99.97%
```

```
05/10/99 11:20 ALARM [1] REPEAT
WARNING: CPU test 99.997%
```

```
05/10/99 11:25 ALARM [1] END
RESET: CPU test 22.86%
EXEC: end.script
```

If you are using a color workstation, the following output is highlighted:

```
CRITICAL statements are RED
```

```
MAJOR statements are MAGENTA
```

```
MINOR statements are YELLOW
```

```
WARNING statements are CYAN
```

```
NORMAL statements are GREEN
```

The next example shows an alarm summary that is displayed after alarm events are listed. The first column lists the alarm number, the second column lists the number of times the alarm condition occurred, and the third column lists the total duration of the alarm condition.

Performance Alarm Summary:

Alarm	Count	Minutes
1	574	2865
2	0	0

Analysis coverage using "alarmdef":

```
Start: 05/04/99 08:00 Stop: 05/06/99 23:59
```

Total time analyzed: Days: 2 Hours: 15 Minutes: 59

Alarm Definition Components

An alarm occurs when one or more of the conditions you define continues over a specified duration. The alarm definition can include an action to be performed at the start or end of the alarm.

A condition is a comparison between two or more items. The compared items can be metric names, constants, or variables. For example:

```
ALARM gbl_cpu_total_util > 95 FOR 5 MINUTES
```

An action can be specified to be performed when the alarm starts, ends, or repeats. The action can be one of the following:

- an ALERT, which sends a message to HPOM or an SNMP trap to NNM
- an EXEC, which performs an operating system command, or
- a PRINT, which sends a message to stdout when processed using the utility program.

For example:

```
ALARM gbl_swap_space_util > 95 FOR 5 MINUTES
  START
    RED ALERT "Global swap space is nearly full"
  END
  RESET ALERT "End of global swap space full condition"
```

You can create more complex actions using Boolean logic, loops through multiple-instance data such as applications, and variables. (For more information, see the next section, Alarm Syntax Reference).

You can also use the INCLUDE statement to identify additional alarm definition files you want used. For example, you may want to break up your alarm definitions into smaller files.

Alarm Syntax Reference

This section describes the statements that are available in the alarm syntax. You may want to look at the `alarmdef` file for examples of how the syntax is used to create useful alarm definitions.

Alarm Syntax

```
ALARM condition [[AND,OR]condition]
  FOR duration [SECONDS, MINUTES]

  [TYPE="string"]
  [SERVICE="string"]
  [SEVERITY=integer]
  [START action]
  [REPEAT EVERY duration [SECONDS, MINUTES] action]
  [END action]
```



```
[RED, CRITICAL, ORANGE, MAJOR, YELLOW, MINOR, CYAN, WARNING,  
GREEN, NORMAL, RESET] ALERT message  
  
EXEC "UNIX command"  
  
PRINT message  
IF condition  
    THEN action  
    [ELSE action]  
  
{APPLICATION, PROCESS, DISK, LVOLUME, TRANSACTION, NETIF, CPU,  
FILESYSTEM} LOOP action  
  
INCLUDE "filename"  
  
USE "data source name"  
  
[VAR] name = value  
  
ALIAS name = "replaced-name"  
  
SYMPTOM variable [ TYPE = {CPU, DISK, MEMORY, NETWORK}]  
    RULE condition PROB probability  
    [RULE condition PROB probability]  
    .  
    .
```

Conventions

- Braces ({}) indicate that one of the choices is required.
- Brackets ([]) indicate an optional item.
- Items separated by commas within brackets or braces are options. Choose only one.
- Italics indicate a variable name that you replace.
- All syntax keywords are in uppercase.

Common Elements

The following elements are used in several statements in the alarm syntax and are described below.

- comments
- compound statements
- conditions
- constants
- expressions
- metric names
- messages

Comments

You can precede comments either by double forward slashes (//) or the pound sign (#). In both cases, the comment ends at the end of the line. For example:

```
# any text or characters
```

or

```
// any text or characters
```

Compound Statements

Compound statements allow a list of statements to be executed as a single statement. A compound statement is a list of statements inside braces ({}). Use the compound statement with the IF statement, the LOOP statement, and the START, REPEAT, and END clauses of the ALARM statement. Compound statements cannot include ALARM and SYMPTOM statements.

```
{  
  statement  
  statement  
}
```

In the example below, `highest_cpu` defines a variable. The `highest_cpu` value is saved and notifies you only when that `highest_cpu` value is exceeded by a higher `highest_cpu` value.

```
highest_cpu = highest_cpu  
IF gbl_cpu_total_util > highest_cpu THEN  
  // Begin compound statement  
  {  
    highest_cpu = gbl_cpu_total_util  
    ALERT "Our new high CPU value is ", highest_cpu, "%"  
  }  
  // End compound statement
```

Conditions

A condition is defined as a comparison between two items.

```
item1 {>, <, >=, <=, ==, !=}item2  
      [AND, OR[item3 {>, <, >=, <=, ==, !=}item4]]
```

where "==" means "equal", and "!=" means "not equal".

Conditions are used in the ALARM, IF, and SYMPTOM statements. An item can be a metric name, a numeric constant, an alphanumeric string enclosed in quotes, an alias, or a variable. When comparing alphanumeric items, only == or != can be used as operators.

Constants

Constants can be either numeric or alphanumeric. An alphanumeric constant must be enclosed in double quotes. For example:

```
345
345.2
"Time is"
```

Constants are useful in expressions and conditions. For example, you may want to compare a metric against a constant numeric value inside a condition to generate an alarm if it is too high, such as

```
gbl_cpu_total_util > 95
```

Expressions

Arithmetic expressions perform one or more arithmetic operations on two or more operands. You can use an expression anywhere you would use a numeric value. Legal arithmetic operators are:

```
+, -, *, /
```

Parentheses can be used to control which parts of an expression are evaluated first.

For example:

```
Iteration + 1
gbl_cpu_total_util - gbl_cpu_user_mode_util
( 100 - gbl_cpu_total_util ) / 100.0
```

Metric Names

When you specify a metric name in an alarm definition, the current value of the metric is substituted. Metric names must be typed exactly as they appear in the metric definition, except for case sensitivity. Metrics definitions can be found in the *Performance Collection Component Dictionary of Operating Systems Performance Metrics*.

It is recommended that you use fully-qualified metric names if the metrics are from a data source other than the SCOPE data source (such as DSI metrics).

The format for specifying a fully qualified metric is:

```
data_source:instance(class):metric_name
```

A global metric in the SCOPE data source requires no qualification. For example:

```
metric_1
```

An application metric, which is available for each application defined in the SCOPE data source, requires the application name. For example,

```
application_1:metric_1
```

For multi-instance data types such as application, process, disk, netif, transaction, lvolume, cpu, and filesystem, you must associate the metric with the data type name, except when using the

LOOP statement. To do this, specify the data type name followed by a colon, and then the metric name. For example, `other_apps:app_cpu_total_util` specifies the total CPU utilization for the application `other_apps`.

Note: When specifying fully qualified multi-instance metrics and using aliases within aliases, if one of the aliases has a class identifier, we recommend you use the syntax shown in this example:

```
alias my_fs="/dev/vg01/lvol1(LVOLUME)"alarm my_fs:LV_SPACE_UTIL > 50 for 5
minutes
```

If you use an application name that has an embedded space, you must replace the space with an underscore (`_`). For example, `application 1` must be changed to `application_1`. For more information on using names that contain special characters, or names where case is significant, see [ALIAS Statement](#).

If you had a disk named "other" and an application named "other", you would need to specify the class as well as the instance:

```
other (disk):metric_1
```

A global metric in an extracted log file (where `scope_extract` is the data source name) would be specified this way:

```
scope_extract:application_1:metric_1
```

A DSI metric would be specified this way:

```
dsi_data_source:dsi_class:metric_name
```

Note: Any metric names containing special characters (such as asterisks) must be aliased before they are specified.

Messages

A message is the information sent by a PRINT or ALERT statement. It can consist of any combination of quoted alphanumeric strings, numeric constants, expressions, and variables. The elements in the message are separated by commas. For example:

```
RED ALERT "cpu utilization=", gbl_cpu_total_util
```

Numeric constants, metrics, and expressions can be formatted for width and number of decimals. *Width* specifies how wide the field should be formatted; *decimals* specifies how many decimal places to use. Numeric values are right-justified. The - (minus sign) specifies left-justification. Alphanumeric strings are always left-justified. For example:

```
metric names [|[-]width[|decimals]]
gbl_cpu_total_util|6|2   formats as '100.00'
(100.32 + 20)|6         formats as '  120'
gbl_cpu_total_util|-6|0  formats as '100  '
gbl_cpu_total_util|10|2  formats as '   99.13'
gbl_cpu_total_util|10|4  formats as '  99.1300'
```

ALARM Statement

The ALARM statement defines a condition or set of conditions and a duration for the conditions to be true. Within the ALARM statement, you can define actions to be performed when the alarm condition starts, repeats, and ends. Conditions or events that you might want to define as alarms include:

- when global swap space has been nearly full for 5 minutes
- when the memory paging rate has been too high for 1 interval
- when your CPU has been running at 75 percent utilization for the last ten minutes

Syntax

```
ALARM condition [[AND,OR]condition]
  FOR duration{SECONDS, MINUTES}
  [TYPE="string"]
  [SERVICE="string"]
  [SEVERITY=integer]
  [START action]
  [REPEAT EVERY duration {SECONDS, MINUTES} action]
  [END action]
```

- The ALARM statement must be a top-level statement. It cannot be nested within any other statement. However, you can include several ALARM conditions in a single ALARM statement. If the conditions are linked by AND, all conditions must be true to trigger the alarm. If they are linked by OR, any one condition will trigger the alarm.
- TYPE is a quoted string of up to 38 characters. If you are sending alarms, you can use TYPE to categorize alarms and to specify the name of a graph template to use.
- SERVICE is a quoted string of up to 200 characters. If you are using ServiceNavigator, you can link your Performance Collection Component alarms with the services you defined in ServiceNavigator (see the *HP Operations ServiceNavigator Concepts and Configuration Guide*).

```
SERVICE="Service_id"
```

- SEVERITY is an integer from 0 to 32767.
- START, REPEAT, and END are *keywords* used to specify what action to take when alarm conditions are met, met again, or stop. You should always have at least one of START, REPEAT, or END in an ALARM statement. Each of these keywords is followed by an *action*.
- *action* – The action most often used with an ALARM START, REPEAT, or END is the ALERT statement. However, you can also use the EXEC statement to mail a message or run a batch file, or a PRINT statement if you are analyzing historical log files with the utility program. Any syntax statement is legal except another ALARM.

START, REPEAT, and END actions can be compound statements. For example, you can use compound statements to provide both an ALERT and an EXEC.

- *Conditions* – A condition is defined as a comparison between two items.

```
item1 {>, <, >=, <=, ==, !=}item2
  [AND, OR[item3 {>, <, >=, <=, ==, !=}item4]]
```

where "==" means "equal", and "!=" means "not equal"

An item can be a metric name, a numeric constant, an alphanumeric string enclosed in quotes, an alias, or a variable. When comparing alphanumeric items, only == or != can be used as operators.

You can use compound conditions by specifying the "OR" and "AND" operator between subconditions. For example:

```
ALARM gbl_cpu_total_util > 90 AND  
gbl_pri_queue > 1 for 5 minutes
```

- You also can use compound conditions *without* specifying the "OR" and "AND" operator between subconditions. For example:

```
ALARM gbl_cpu_total_util > 90  
gbl_cpu_sys_mode_util > 50 for 5 minutes
```

will cause an alarm when both conditions are true.

FOR *duration* SECONDS, MINUTES specifies the time period the condition must remain true to trigger an alarm.

Use caution when specifying durations of less than one minute, particularly when there are multiple data sources on the system. Performance can be seriously degraded if each data source must be polled for data at very small intervals. The duration must be a multiple of the longest collection interval of the metrics mentioned in the alarm condition.

Do not set a duration that is shorter than the collection interval. Since the metric value does not change until the next collection cycle is complete, setting a duration shorter than the collection interval means unnecessary processing for generating duplicate alarms.

In the `parm` file, the following parameters under the `collectioninterval` parameter control the collection interval:

`global`: This parameter indicates the collection interval (in seconds) for all metrics that are logged into the scope and `dsi` log files except for the process metrics. By default, this is set to five minutes.

`process`: This parameter indicates the collection interval (in seconds) for process metrics.

For more information about the `collectioninterval` parameter, see [Configure Data Logging Intervals](#).

- REPEAT EVERY*duration*SECONDS, MINUTES specifies the time period before the alarm is repeated.

How It Is Used

The alarm cycle begins on the first interval that all of the ANDed, or one of the ORed alarm conditions have been true for at least the specified duration. At that time, the alarm generator executes the *START action*, and on each subsequent interval checks the REPEAT condition. If enough time has transpired, the *action* for the REPEAT clause is executed. (This continues until one or more of the alarm conditions becomes false.) This completes the alarm cycle and the END statement is executed if there is one.

In order to be notified of the alarm, use the ALERT statement within the START and END statements. If you do not specify an END ALERT, the alarm generator automatically sends an alarm to HPOM and sends an SNMP trap to NNM.

Examples

The following ALARM example sends a red alert when the swap utilization is high for 5 minutes. It is similar to an alarm condition in the default `alarmdef` file. Do not add this example to your `alarmdef` file without removing the default alarm condition, or your subsequent alert messages may be confusing.

```
ALARM gbl_swap_space_util > 90 FOR 5 MINUTES
  START
    RED ALERT "swap utilization is very high "
  REPEAT EVERY 15 MINUTES
    RED ALERT "swap utilization is still very high "
  END
  RESET ALERT "End of swap utilization condition"
```

This ALARM example tests the metric `gbl_swap_space_util` to see if it is greater than 90. Depending on how you configured the alarm generator, the ALERT can be sent to NNM via an SNMP trap or as a message to Operations Manager.

The REPEAT statement checks for the `gbl_swap_space_util` condition every 15 minutes. As long as the metric remains greater than 90, the REPEAT statement will send the message “swap utilization is still very high” every 15 minutes.

When the `gbl_swap_space_util` condition goes below 90, the RESET ALERT statement with the “End of swap utilization condition” message is sent.

The following example defines a compound action within the ALARM statement. This example shows you how to cause a message to be mailed when an event occurs.

```
ALARM gbl_cpu_total_util > 90 FOR 5 MINUTES
  START
  {
    RED ALERT "Your CPU is busy."
    EXEC "echo 'cpu is too high'| mailx root"
  }
  END
  RESET ALERT "CPU no longer busy."
```

The ALERT can trigger an SNMP trap to be sent to NNM or a message to be sent to HPOM. The EXEC can trigger a mail message to be sent as a local action on your node, depending on how you configured your alarm generator.

The following two examples show the use of multiple conditions. You can have more than one test condition in the ALARM statement. In this case, each statement must be true for the ALERT to be sent.

The following ALARM example tests the metric `gbl_cpu_total_util` and the metric `gbl_cpu_sys_mode_util`. If both conditions are true, the RED ALERT statement sends a red alert. When either test condition becomes false, the RESET is sent.

```
ALARM gbl_cpu_total_util > 90
  AND gbl_cpu_sys_mode_util > 50 FOR 5 MINUTES
  START
  RED ALERT "CPU busy and Sys Mode CPU util is high."
```

```
END
  RESET ALERT "The CPU alert is now over."
```

The next ALARM example tests the metric `gbl_cpu_total_util` and the metric `gbl_cpu_sys_mode_util`. If either condition is true, the RED ALERT statement sends a red alert.

```
ALARM gbl_cpu_total_util > 90
  OR
    gbl_cpu_sys_mode_util > 50 FOR 10 MINUTES
START
  RED ALERT "Either total CPU util or sys mode CPU high"
```

Do not use metrics that are logged at different intervals in the same alarm. For example, you should not loop on a process (logged at 1-minute intervals) based on the value of a global metric (logged at 5-minute intervals) in a statement like this:

```
IF global_metric THEN  PROCESS LOOP...
```

The different intervals cannot be synchronized as you might expect, so results will not be valid.

Note: For GlancePlus, use the process metrics inside a process loop in order to send alarm for all the processes.

ALERT Statement

The ALERT statement allows a message to be sent to Network Node Manager or Operations Manager. The ALERT statement is most often used as an action within an ALARM. It could also be used within an IF statement to send a message as soon as a condition is detected instead of after the duration has passed. If an ALERT is used outside of an ALARM or IF statement, the message will be sent at every interval.

Syntax

```
[RED, CRITICAL, ORANGE, MAJOR, YELLOW, MINOR, CYAN, WARNING, GREEN, NORMAL, RESET]
ALERT message
```

- RED is synonymous with CRITICAL, ORANGE is synonymous with MAJOR, YELLOW is synonymous with MINOR, CYAN is synonymous with WARNING, and GREEN is synonymous with NORMAL. These keywords turn the alarm symbol to the color associated with the alarm condition.
- RESET — Sends a RESET ALERT with a message when the ALARM condition ends. If you have not defined a reset in the alarm definition, sends a RESET ALERT without a message when the ALARM condition ends.
- *message* — A combination of strings and numeric values used to create a message. Numeric values can be formatted with the parameters `[| [-]width[|decimals]]`. *Width* specifies how wide the field should be formatted; *decimals* specifies how many decimal places to use. Numeric values are right-justified. The - (minus sign) specifies left-justification. Alphanumeric strings are always left-justified.

How It Is Used

The ALERT can also trigger an SNMP trap to be sent to NNM or a message to be sent to HPOM, depending on how you configured your alarm generator. For alert messages sent to HPOM, the WARNINGS will be displayed in blue in the message browser.

Example

An typical ALERT statement is:

```
RED ALERT "CPU utilization = ", gbl_cpu_total_util
```

If you have Network Node Manager, this statement creates a critical severity alarm in the Alarm Browser window in Network Node Manager.

EXEC Statement

The EXEC statement allows you to specify a system (UNIX or Windows) command to be performed on the local system. For example, you could use the EXEC statement to send mail to an IT administrator each time a certain condition is met.

EXEC should be used within an ALARM or IF statement so the command is performed only when specified conditions are met. If an EXEC statement is used outside of an ALARM or IF statement, the action will be performed at unpredictable intervals.

Syntax

```
EXEC "system command"
```

system command — a command to be performed on the local system.

Do not use embedded double quotes (") in EXEC statements. Doing so causes `perfalarm` to fail to send the alarm to HPOM. Use embedded single (') quotes instead. For example:

```
EXEC "echo 'performance problem detected' "
```

```
EXEC "mkdir c:\\directory\\filename"
```

The syntax of the EXEC statement requires the path name of the file to be enclosed within double quotes. However, if a path name contains spaces, the path name must be enclosed within single quotes, which must be again enclosed within double quotes.

Example:

```
EXEC "'C:\\Program Files\\Mail Program\\SendMail.exe'"
```

If any arguments to the system command of the EXEC statement contains single quotes, the program must be enclosed within single quotes as the first pair of single quotes (') are converted into double quotes (") while you run the command with the EXEC statement.

Example:

```
EXEC "'echo' 'test execution'"
```

In the above example, echo is the program enclosed within single quotes as it contains an argument (in this case, test execution) with single quotes. Furthermore, as per the syntax of the EXEC statement, the entire string of the command must be enclosed in double quotes.

Do not use embedded double quotes (") in EXEC statements; perfalarm will fail to send the alarm to the HPOM. Use embedded single quotes (') instead.

For example:

```
EXEC "'echo' 'dialog performance problem'"
```

In the above example, echo is the program enclosed within single quotes as it contains an argument (in this case, dialog performance problem) with single quotes. Further, as per the syntax of the EXEC statement, the entire string of the command must be enclosed in double quotes.

How It Is Used

The EXEC can trigger a local action on your local system, depending on how you configured your alarm generator. For example, you can turn local actions on or off. If you configured your alarm generator to send information to HPOM local actions will not usually be performed.

Examples

In the following example, the EXEC statement performs the UNIX mailx command when the gbl_disk_util_peak metric exceeds 20.

```
IF gbl_disk_util_peak > 20 THEN
    EXEC "echo 'high disk utilization detected'| mailx root"
```

The next example shows the EXEC statement sending mail to the system administrator when the network packet rate exceeds 1000 per second average for 15 minutes.

```
ALARM gbl_net_packet_rate > 1000 for 15 minutes
    TYPE = "net busy"
    SEVERITY = 5
    START
    {
        RED ALERT "network is busy"
        EXEC "echo 'network busy condition detected'| mailx root"
    }
    END
    RESET ALERT "NETWORK OK"
```

Note: Be careful when using the EXEC statement with commands or scripts that have high overhead if it will be performed often.

The alarm generator executes the command and waits until it completes before continuing. We recommend that you not specify commands that take a long time to complete.

PRINT Statement

The PRINT statement allows you to print a message from the utility program using its analyze function. The alarm generator ignores the PRINT statement.

Syntax

PRINT *message*

- *message* — A combination of strings and numeric values that create a message. Numeric values can be formatted with the parameters `[[-]width [|decimals]]`. *Width* specifies how wide the field should be formatted; *decimals* specifies how many decimal places to use. Alphanumeric components of a message must be enclosed in quotes. Numeric values are right-justified. The - (minus sign) specifies left-justification. Alphanumeric strings are always left-justified.

Example

```
PRINT "The total time the CPU was not idle is",  
      gbl_cpu_total_time |6|2, "seconds"
```

When executed, this statement prints a message such as the following:

```
The total time the CPU was not idle is 95.00 seconds
```

IF Statement

Use the IF statement to define a condition using IF-THEN logic. The IF statement should be used within the ALARM statement. However, it can be used by itself or any place in the alarmdef file where IF-THEN logic is needed.

If you specify an IF statement outside of an ALARM statement, you do not have control over how frequently it gets executed.

Syntax

IF *condition* THEN *action* [ELSE *action*]

IF *condition* — A condition is defined as a comparison between two items.

```
item1 {>, <, >=, <=, ==, !=}item2  
  [AND, OR[item3 {>, <, >=, <=, ==, !=}item4]]
```

where "==" means "equal", and "!=" means "not equal".

An item can be a metric name, a numeric constant, an alphanumeric string enclosed in quotes, an alias, or a variable. When comparing alphanumeric strings, only == or != can be used as operators.

action — Any action, or set a variable. (ALARM is not valid in this case.)

How It Is Used

The IF statement tests the *condition*. If the *condition* is true, the *action* after the THEN is executed. If the *condition* is false, the *action* depends on the optional ELSE clause. If an ELSE clause has been specified, the *action* following it is executed; otherwise the IF statement does nothing.

Example

In this example, a CPU bottleneck symptom is calculated and the resulting bottleneck probability is used to define cyan or red ALERTs. Depending on how you configured your alarm generator, the ALERT triggers an SNMP trap to NNM or the message "End of CPU Bottleneck Alert" to Operations Manager along with the percentage of CPU used.

```
SYMPTOM CPU_Bottleneck > type=CPU
  RULE gbl_cpu_total_util > 75 prob 25
  RULE gbl_cpu_total_util > 85 prob 25
  RULE gbl_cpu_total_util > 90 prob 25
  RULE gbl_cpu_total_util > 4 prob 25

ALARM CPU_Bottleneck > 50 for 5 minutes
  TYPE="CPU"
  START
    IF CPU_Bottleneck > 90 then
      RED ALERT "CPU Bottleneck probability= ",
        CPU_Bottleneck, "%"
    ELSE
      CYAN ALERT "CPU Bottleneck probability= ",
        CPU_Bottleneck, "%"
  REPEAT every 10 minutes
    IF CPU_Bottleneck > 90 then
      RED ALERT "CPU Bottleneck probability= ",
        CPU_Bottleneck, "%"
    ELSE
      CYAN ALERT "CPU Bottleneck probability= ",
        CPU_Bottleneck, "%"
  END
  RESET ALERT "End of CPU Bottleneck Alert"
```

Do not use metrics that are logged at different intervals in the same statement. For instance, you should not loop on a process (logged at 1-minute intervals) based on the value of a global metric (logged at 5-minute intervals) in a statement like this:

```
IF global_metric THEN PROCESS LOOP ...
```

The different intervals cannot be synchronized as you might expect, so results will not be valid.

LOOP Statement

The LOOP statement goes through multiple-instance data types and performs the *action* defined for each instance.

Syntax

```
{APPLICATION, PROCESS, LVOLUME, DISK, CPU, FILESYSTEM, TRANSACTION, NETIF, LOGICAL}  
LOOP
```

action

- APPLICATION, PROCESS, LVOLUME, DISK, CPU, FILESYSTEM, TRANSACTION, NETIF, LOGICAL — Performance Collection Component data types that contain multi-instance data.
- *action* — PRINT, EXEC, ALERT, set variables.

How It Is Used

As LOOP statements iterate through each instance of the data type, metric values change. For instance, the following LOOP statement prints the name of each application to `stdout` if you are using the utility program's `analyze` command.

```
APPLICATION LOOP
```

```
PRINT app_name
```

A LOOP can be nested within another LOOP statement up to a maximum of five levels.

In order for the LOOP to execute, the LOOP statement must refer to one or more metrics of the same data type as the type defined in the LOOP statement.

Example

You can use the LOOP statement to cycle through all active applications.

The following example shows how to determine which application has the highest CPU at each interval.

```
highest_cpu = 0  
APPLICATION loop  
  IF app_cpu_total_util > highest_cpu THEN  
  {  
    highest_cpu = app_cpu_total_util  
    big_app = app_name  
  }  
  
  ALERT "Application ", app_name, " has the highest cpu util at ",highest_cpu_  
util|5|2, "%"  
  
  ALARM highest_cpu > 50  
  START
```

```
    RED ALERT  big_app, " is the highest CPU user at ", highest_cpu, "%"
REPEAT EVERY 15 minutes
    CYAN ALERT big_app, " is the highest CPU user at ", highest_cpu, "%"
END
RESET ALERT "No applications using excessive cpu"
```

INCLUDE Statement

Use the `INCLUDE` statement to include another alarm definitions file along with the default `alarmdef` file.

Syntax

```
INCLUDE "filename"
```

where *filename* is the name of another alarm definitions file. The file name must always be fully qualified.

How It Is Used

The `INCLUDE` statement could be used to separate logically distinct sets of alarm definitions into separate files.

Example

For example, if you have some alarm definitions in a separate file for your transaction metrics and it is named

```
trans_alarmdef1
```

You can include it by adding the following line to the alarm definitions in your `alarmdef1` file:

```
INCLUDE "/var/opt/perf/trans_alarmdef1"
```

USE Statement

You can add the `USE` statement to simplify the use of metric names in the `alarmdef` file when data sources other than the default `SCOPE` data source are referenced. This allows you to specify a metric name without having to include the data source name.

The data source name must be defined in the `datasources` file. The `alarmdef` file will fail its syntax check if an invalid or unavailable data source name is encountered.

Note: The appearance of a `USE` statement in the `alarmdef` file does not imply that all metric names that follow will be from the specified data source.

Syntax

USE "*datasourcename*"

How It Is Used

As the alarm generator checks the `alarmdef` file for valid syntax, it builds an ordered search list of all data sources that are referenced in the file. `Perfalarm` sequentially adds entries to this data source search list as it encounters fully-qualified metric names or `USE` statements. This list is subsequently used to match metric names that are not fully qualified with the appropriate data source name. The `USE` statement provides a convenient way to add data sources to `perfalarm`'s search list, which then allows for shortened metric names in the `alarmdef` file. For a discussion of metric name syntax, see [Metric Names](#) earlier in this chapter.

`Perfalarm`'s default behavior for matching metric names to a data source is to look first in the `SCOPE` data source for the metric name. This implied `USE "SCOPE"` statement is executed when `perfalarm` encounters the first metric name in the `alarmdef` file. This feature enables a default search path to the `SCOPE` data source so that `SCOPE` metrics can be referenced in the `alarmdef` file without the need to fully qualify them. This is shown in the following example on the next page.

```
ALARM gbl_cpu_total_util > 80 FOR 10 MINUTES
    START RED ALERT "CPU utilization too high"

USE "ORACLE7"

ALARM ActiveTransactions >= 95 FOR 5 MINUTES
    START RED ALERT "Nearing limit of transactions for ORACLE7"
```

When `perfalarm` checks the syntax of the `alarmdef` file containing the above statements, it encounters the metric `"gbl_cpu_total_util"` and then tries to find its data source. `Perfalarm` does not yet have any data sources in its search list of data sources, so it executes an implied `USE "SCOPE"` statement and then searches the `SCOPE` data source to find the metric name. A match is found and `perfalarm` continues checking the rest of the `alarmdef` file.

When `perfalarm` encounters the `USE "ORACLE7"` statement, it adds the `ORACLE7` data source to the search list of data sources. When the `"ActiveTransactions"` metric name is encountered, `perfalarm` sequentially searches the list of data sources starting with the `SCOPE` data source. `SCOPE` does not contain that metric name, so the `ORACLE7` data source is searched next and a match is found.

If `perfalarm` does not find a match in any data source for a metric name, an error message is printed and `perfalarm` terminates.

To change the default search behavior, a `USE` statement can be added to the beginning of the `alarmdef` file before any references to metric names. This will cause the data source specified in the `USE` statement to be added to the search list of data sources before the `SCOPE` data source. The data source (s) in the `USE` statement(s) will be searched before the `SCOPE` data source to find matches to the metrics names. This is shown in the following example.

Once a data source has been referenced with a `USE` statement, there is no way to change its order or to remove it from the search list.

```
USE "ORACLE7"
```

```
ALARM gbl_cpu_total_util > 80 FOR 10 MINUTES  
    START RED ALERT "CPU utilization too high"
```

```
ALARM ActiveTransactions >= 95 FOR 5 MINUTES  
    START RED ALERT "Nearing limit of  
    transactions for ORACLE7"
```

In the example above, the order of the statements in the `alarmdef` file has changed. The `USE "ORACLE7"` statement is defined before any metric names are referenced, therefore the `ORACLE7` data source is added as the first data source in the search list of data sources. The implied `USE "SCOPE"` statement is executed when `perfalarm` encounters the first metric name `"gbl_cpu_total_util."` Because the `"gbl_cpu_total_util"` metric name is not fully-qualified, `perfalarm` sequentially searches through the list of data sources starting with `ORACLE7`. `ORACLE7` does not contain that metric name so the `SCOPE` data source is searched next and a match is found.

`Perfalarm` continues checking the rest of the `alarmdef` file. When `perfalarm` encounters the `"ActiveTransactions"` metric, it sequentially searches the list of data sources starting with `ORACLE7`. A match is found and `perfalarm` continues searching the rest of the `alarmdef` file. If `perfalarm` does not find a match in any data source for a metric name (that is not fully-qualified), an error message will be printed and `perfalarm` terminates.

Be careful how you use the `USE` statement when multiple data sources contain the same metric names. `Perfalarm` sequentially searches the list of data sources. If you are defining alarm conditions from different data sources that use the same metric names, you must qualify the metric names with their data source names to guarantee that the metric value is retrieved from the correct data source. This is shown in the following example where the metric names in the alarm statements each include their data sources.

```
ALARM ORACLE7:ActiveTransactions >= 95 FOR 5 MINUTES  
    START RED ALERT "Nearing limit of transactions for ORACLE7"
```

```
ALARM FINANCE:ActiveTransactions >= 95 FOR 5 MINUTES  
    START RED ALERT "Nearing limit of transactions for FINANCE"
```

VAR Statement

The `VAR` statement allows you to define a variable and assign a value to it.

Syntax

```
[VAR] name = value
```

name — Variable names must begin with a letter and can include letters, digits, and the underscore character. Variable names are not case-sensitive.

value — If the value is an alphanumeric string, it must be enclosed in quotes.

How It Is Used

`VAR` assigns a value to the user variable. If the variable did not previously exist, it is created.

Once defined, variables can be used anywhere in the `alarmdef` file.

Examples

You can define a variable by assigning something to it. The following example defines the numeric variable *highest_CPU_value* by assigning it a value of zero.

```
highest_CPU_value = 0
```

The next example defines the alphanumeric variable *my_name* by assigning it an empty string value.

```
my_name = ""
```

ALIAS Statement

The ALIAS statement allows you to substitute an alias if any part of a metric name (class, instance, or metric) has a case-sensitive name or a name that includes special characters. These are the only circumstances where the ALIAS statement should be used.

Syntax

```
ALIAS name = "replaced-name"
```

- *name* — The name must begin with a letter and can include letters, digits, and the underscore character.
- *replaced-name* — The name that must be replaced by the ALIAS statement to make it uniquely recognizable to the alarm generator.

How It Is Used

Because of the way the `alarmdef` file is processed, if any part of a metric name (class, instance, or metric name) can be identified uniquely only by recognizing uppercase and lowercase, you will need to create an alias. You will also need to create an alias for any name that includes special characters. For example, if you have applications called "BIG" and "big," you'll need to alias "big" to ensure that they are viewed as different applications. You must define the alias somewhere in the `alarmdef` file before the *first* instance of the name you want substituted.

Examples

Because you cannot use special characters or upper and lower case in the syntax, using the application name "AppA" and "appa" could cause errors because the processing would be unable to distinguish between the two. You would alias "AppA" to give it a uniquely recognizable name. For example:

```
ALIAS appa_uc = "AppA"  
ALERT "CPU alert for AppA.util is", appa_uc: app_cpu_total_util
```

If you are using an alias for an instance with a class identifier, include both the instance name and the class name in the alias. The following example shows the alias for the instance name 'other' and the class name 'APPLICATION.'

```
ALIAS my_app="other(APPLICATION)"  
ALERT my_app:app_cpu_total_util > 50 for 5 minutes
```

SYMPTOM Statement

A SYMPTOM provides a way to set a single variable value based on a set of conditions. Whenever any of the conditions is true, its probability value is added to the value of the SYMPTOM variable.

Syntax

*SYMPTOM**variable*

*RULE**condition* *PROB* *probability*

[*RULE* *condition* *PROB* *probability*]

.
. .
.

- The keywords SYMPTOM and RULE are used exclusively in the SYMPTOM statement and cannot be used in other syntax statements. The SYMPTOM statement must be a top-level statement and cannot be nested within any other statement. No other statements can follow SYMPTOM until all its corresponding RULE statements are finished.
- *variable* is a variable name that will be the name of this symptom. Variable names defined in the SYMPTOM statement can be used in other syntax statements, but the variable value should not be changed in those statements.
- *RULE* is an option of the SYMPTOM statement and cannot be used independently. You can use as many *RULE* options as needed within the SYMPTOM statement. The SYMPTOM variable is evaluated according to the rules at each interval.
- *condition* is defined as a comparison between two items.
item1 {>, <, >=, <=, ==, !=}item2
[item3 {>, <, >=, <=, ==, !=}item4]
where "==" means "equal" and "!=" means "not equal".
An item can be a metric name, a numeric constant, an alphanumeric string enclosed in quotes, an alias, or a variable. When comparing alphanumeric items, only == or != can be used as operators.
- *probability* is a numeric constant. The probabilities for each true SYMPTOM RULE are added together to create a SYMPTOM value.

How It Is Used

The sum of all probabilities where the condition between measurement and value is true is the probability that the symptom is occurring.

Example

```
SYMPTOM CPU_Bottleneck
RULE gbl_cpu_total_util > 75  PROB 25
RULE gbl_cpu_total_util > 85  PROB 25
RULE gbl_cpu_total_util > 90  PROB 25
RULE gbl_run_queue      > 3   PROB 50
IF CPU bottleneck > 50 THEN
CYAN ALERT "The CPU symptom is: ", CPU_bottleneck
```

Alarm Definition Examples

The following examples show typical uses of alarm definitions.

Example of a CPU Problem

Depending on how you configured the alarm generator, this example triggers an SNMP trap to Network Node Manager or a message to Operations Manager whenever CPU utilization exceeds 90 percent for 5 minutes and the CPU run queue exceeds 3 for 5 minutes.

```
ALARM gbl_cpu_total_util > 90 AND
      gbl_run_queue > 3 FOR 5 MINUTES
START
  CYAN ALERT "CPU too high at", gbl_cpu_total_util, "%"
REPEAT EVERY 20 MINUTES
{
  RED ALERT "CPU still to high at ", gbl_cpu_total_util, "%"
  EXEC "/usr/bin/pager -n 555-3456"
}
END
  RESET ALERT "CPU at ", gbl_cpu_total_util, "% - RELAX"
```

If both conditions continue to hold true after 20 minutes, a critical severity alarm can be created in NNM. A program is then run to page the system administrator.

When either one of the alarm conditions fails to be true, the alarm symbol is deleted and a message is sent showing the global CPU utilization, the time the alert ended, and a note to RELAX.

Example of Swap Utilization

In this example, depending on how you configured the alarm generator, the ALERT can trigger an SNMP trap to be sent to NNM or a message to be sent to HPOM, whenever swap space utilization exceeds 95 percent for 5 minutes.

```
ALARM gbl_swap_space_util > 95 FOR 5 MINUTES
START
  RED ALERT "GLOBAL SWAP space is nearly full "
```

```
END
  RESET ALERT "End of GLOBAL SWAP full condition"
```

Example of Time-Based Alarms

You can specify a time interval during which alarm conditions can be active. For example, if you are running system maintenance jobs that are scheduled to run at regular intervals, you can specify alarm conditions for normal operating hours and a different set of alarm conditions for system maintenance hours.

In this example, the alarm will only be triggered during the day from 8:00AM to 5:00PM.

```
start_shift = "08:00"
end_shift = "17:00"

ALARM gbl_cpu_total_util > 80
  TIME > start_shift
  TIME < end_shift for 10 minutes
  TYPE = "cpu"
  START
    CYAN ALERT "cpu too high at ", gbl_cpu_total_util, "%"
  REPEAT EVERY 10 minutes
    RED ALERT "cpu still too high at ", gbl_cpu_total_util, "%"
  END
  IF time == end_shift then
  {
  IF gbl_cpu_total_util > 80 then
    RESET ALERT "cpu still too high, but at the end of shift"
  ELSE
    RESET ALERT "cpu back to normal"
  ELSE
```

Example of Disk Instance Alarms

Alarms can be generated for a particular disk by identifying the specific disk instance name and corresponding metric name.

The following example of alarm syntax generates alarms for a specific disk instance. Aliasing is required when special characters are used in the disk instance.

```
ALIAS diskname=""
ALARM diskname:bydisk_phys_read > 1000 for 5 minutes
TYPE="Disk"
  START
    RED ALERT "Disk "
  REPEAT EVERY 10 MINUTES
    CYAN ALERT "Disk cyan alert"
  END
    RESET ALERT "Disk reset alert"
```

Customizing Alarm Definitions

You specify the conditions that generate alarms in the alarm definitions file `alarmdef`. When Performance Collection Component is first installed, the `alarmdef` file contains a set of default alarm definitions. You can use these default alarm definitions or customize them to suit your needs.

You can customize your `alarmdef` file as follows:

1. Revise your alarm definition(s) as necessary. You can look at examples of the alarm definition syntax elsewhere in this chapter.
2. Save the file.
3. Validate the alarm definitions using the Performance Collection Component utility program:
 - a. Type `utility`
 - b. At the prompt, type `checkdef`

This checks the alarm syntax and displays errors or warnings if there any problems with the file.

4. In order for the new alarm definitions to take effect, type:

```
ovpa restart alarm
```

or

```
mwa restart alarm
```

This causes the alarm generator to stop, restart, and read the customized `alarmdef` file.

You can use a unique set of alarm definitions for each Performance Collection Component system, or you can choose to standardize monitoring of a group of systems by using the same set of alarm definitions across the group.

If the `alarmdef` file is very large, the Performance Collection Component alarm generator and `utility` programs may not work as expected. This problem may or may not occur based on the availability of system resources.

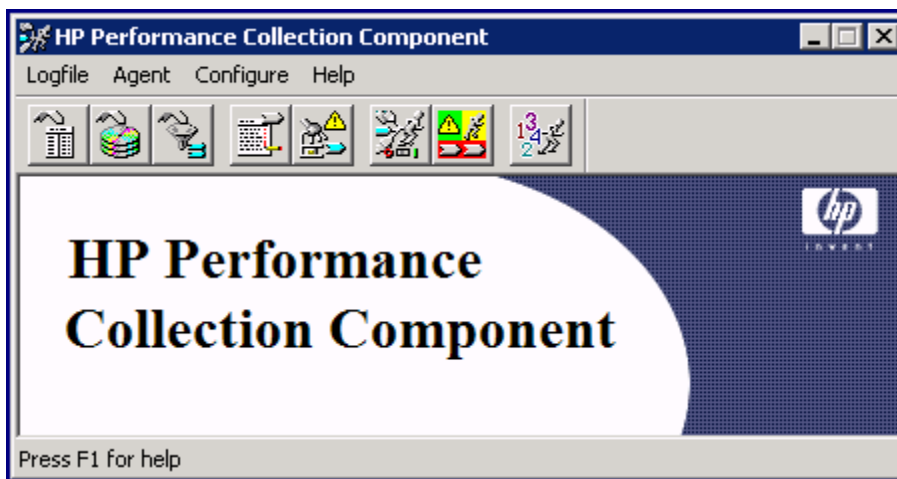
The best way to learn about performance alarms is to experiment with adding new alarm definitions or changing the default alarm definitions.

Chapter 12: Using the Performance Collection Component on Windows

To access the Performance Collection Component graphical user interface click the HP Operations Agent Software icon in the following folder:

Start → Programs → HP → Operations Agent → Performance Collection Component

Performance Collection Component Main Window



This chapter describes the following tasks that you perform using the Performance Collection Component graphical interface:

[Data Types and Classes](#)

[Summarization Levels](#)

[Ranges of Data to Extract or Export](#)

[Extracting Log File Data and Exporting Log File Data](#)

[Archiving Log File Data](#)

[Resizing of Log File](#)

[Scanning a Log File](#)

[Analyzing a Log File](#)

[Configuring Export Templates](#)

[Configuring User Options](#)

[Configuring Collection Parameters](#)

[Configuring Alarm Definitions](#)

[Checking Status](#)

Building Collections of Performance Counters

Before you start using Performance Collection Component for tasks that involve extracting, exporting, or archiving data, read the following sections. These sections discuss the selection of data types, summarization levels, and ranges of the data to be extracted, exported, or archived.

Data Types and Classes

The following types of scoped log file data can be selected for extraction or exporting:

Type of Data	Type of Measurement
global	system-wide, or global information
application	processes in each user-defined application
configuration	system configuration usage
process	selected interesting processes
disk	disk devices usage
filesystem	logical disks usage
logicalsystem	logical system usage
cpu	CPU usage
netif	network interface devices usage
transaction	transaction tracking data

DSI log file data can be selected for exporting according to class. Each class represents one source of incoming data and consists of a group of related data items (metrics) that are logged together.

Summarization Levels

To export data, you must specify the level of summarization you want – detail, summary, or both – when exporting log file data.

Detail specifies that detail data from a 5-minute period is exported from all data types except process, from which detail data from a 1-minute period is exported.

Summary specifies that data summarized over a 1-hour period is exported.

Detail and summary together provide a maximum amount of exported data.

Summarization affects the size of the exported data. For example, hourly summary data is about one-tenth the size of 5-minute detail data.

Ranges of Data to Extract or Export

You can select specific data to extract or export depending on the date and time it was logged. For example, you might want data that was logged every day (Monday through Sunday) from 8:00 am to 8:00 pm during a 30-day period starting at a specific date.

If you do not specify a specific range of data to be exported, data is extracted or exported using the default starting date – the date 30 days before the last date in the log file. Or, if less than 30 days of data is present, the date of the earliest record in the log file.

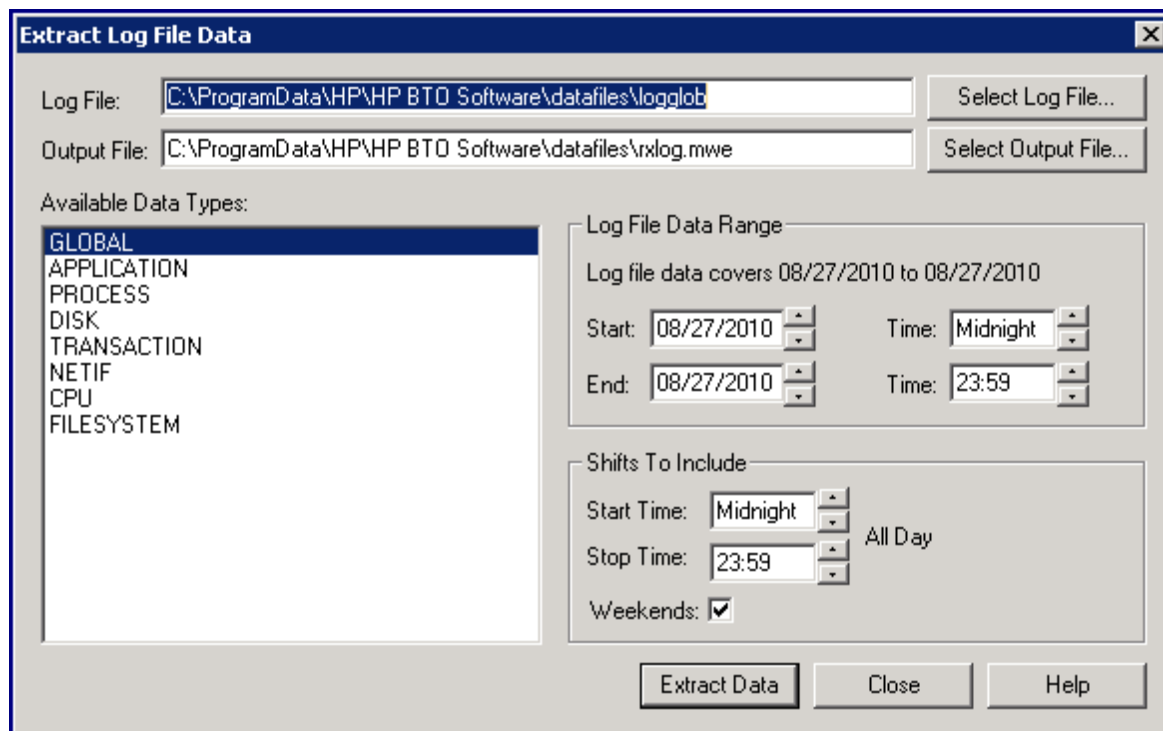
You can also limit extraction or export to data that was logged during specific hours of the day that correspond to work shifts (such as 8:00 am to 5:00 pm) from Monday through Friday. If no shift is specified, the default is extraction or export of 24-hours' worth of data for every day including weekends.

Extracting Log File Data

The data collector, `scopent`, continuously collects data and logs it into raw log files. You can extract specific data from the default global log file set, **logglob**, into extracted log files that can later be used for archiving or for analyzing by analysis programs such as HP Performance Manager. You can also extract data from existing extracted log files. You *cannot* extract DSI log file data.

When you specify **logglob**, all other raw log files in the log file set automatically open. For example, it is not necessary to open the **logproc** log file to extract process data; opening **logglob** enables you to access all data types in the raw log file set.

Extract Log File Data Dialog Box



Note: When you display the Extract Log File Data or Export Log File Data dialog boxes after starting Performance Collection Component, the name of the default global log file, **logglob**, appears in the Log File box to indicate the currently active log file. **Logglob** remains active until you close Performance Collection Component or select a different log file. When you select a different log file, that file's name appears in the Log File box as the currently active log file.

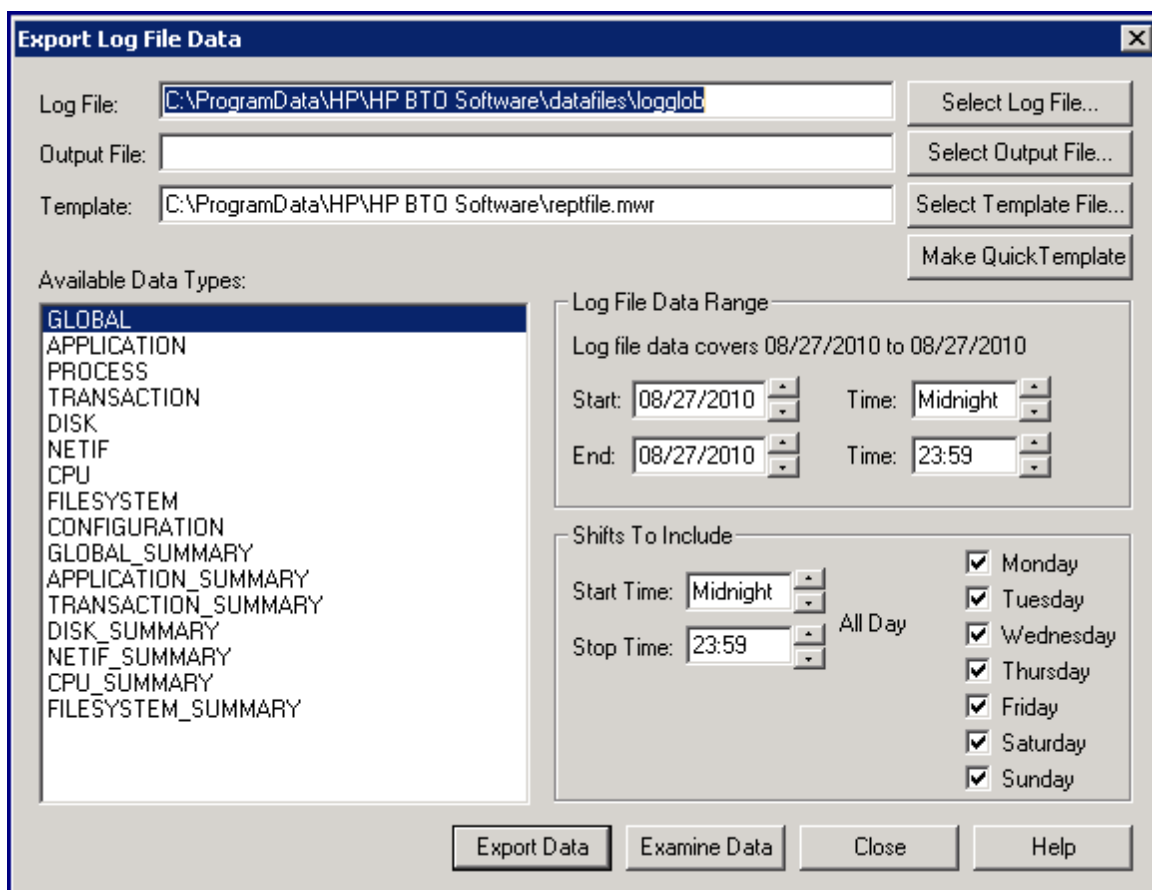
Exporting Log File Data

You can export specific data from raw or extracted scopent log files or from DSI log files into formatted export files that can be used by spreadsheets and other reporting tools.

The export function does *not* remove any data from the log file.

The following sections discuss:

- File attributes that you can specify for the exported data
- Export templates you can use to simplify the export task.
- Default export file
- Overview of the export task
- Export Log File Data Dialog Box



File Attributes

You can assign various file attributes to your exported data, including file formats, values that represent missing data, field separators, column headings, number of minutes for summary intervals, layout, data types, and specific metrics to be included in the export file. These attributes can be saved in export template files or specified directly using the Make Quick Template dialog box. (For more information, see [Making a Quick Export Template.](#))

File Format

The output format for an exported file can be ASCII, datafile, binary, or WK1 (spreadsheet):

ASCII format is printable character data, suitable for printed reports or post-processing by user-written programs or batch files.

Datafile format is similar to ASCII except that all non-numeric items are enclosed in double quotes (" "). Datafile format is useful for transferring data to most spreadsheets and graphic packages.

Binary format is not printable. It is more compact because numeric values are represented as binary integers. It is the most suitable format for input into user-written analysis programs because it needs the least conversion and maintains the highest metric accuracy.

WK1 (spreadsheet) format is compatible with Microsoft Excel, and other spreadsheet, database, and graphing products.

Missing Value

An exported file can contain the data value that replaces data missing from the source log file. Missing data can occur when a metric is not available for certain versions of the scopent collector. In addition, the multiple layout export formats for applications, disks, and transactions reserve space in the output record for every application, disk, or transaction. If no data was logged for a particular entry during an interval, its data will be "missing".

Field Separators

A field separator character can be inserted between each metric in ASCII and datafile format exported files. Separator characters can be printable or non-printable (such as a tab character).

The default separator character is a blank space, but many programs require a comma as a field separator.

Summary Minutes

The number of minutes for each summary interval can be specified. The number of minutes can range from five to 1440 minutes (one day).

Headings

Column headings can be included in exported files. The first record in the file is exported data (except in WK1 format files). However, if you include headings in the file, ASCII and datafile format files have the export file title (if specified) plus column headings for each column of metrics written *before* the first data records in the file. Headings in binary format files are written before the first record in the file and contain descriptions of the metrics.

WK1 files always contain headings.

Multiple Layout

You can specify multiple layouts (per record output) for multi-instance data types such as application or disk.

Single layout writes one record for every application or disk that was active during the time interval. Multiple layout writes one record for each time interval, with part of that record reserved for each configured application or disk.

Export File Title

You can specify the title for an export file. The title can contain literal strings as well as substitution keywords. The following items can be substituted in the export title string:

!date	the date the export file is created
!time	the time of day the export file is created
!logfile	the name of the log file from which data is obtained
!collector	the name and version of the collector program (either scopent or dsilog)
!class	the type of data requested
!system_id	the identifier of the system that collects the scopent raw or extracted log file data (not valid with DSI log file data)

For example, you could type the following string:

```
export "!system_id data from !logfile on !date !time"
```

The string would generate a title similar to the following:

```
gemini data from logglob on 10/25/99 08:30 AM
```

Export File Templates

The export task uses export templates that define the file attributes for your exported file. The default file attributes are taken from the file `<rpmttools>\data\reptfile.mwr` that specifies:

ASCII file format

A 0 (zero) for the missing value

A blank space as the field separator

60-minute summaries

Headings are included

A recommended set of metrics for a given data type or class is included in the export

You can either specify a different export template file or make ad hoc file attribute specifications (see [Making a Quick Export Template](#))

You can also create customized export templates using the Configure Export Templates dialog box (see [Configuring Export Templates](#))

Default Export Files

If you do not specify an output file to contain your exported data, the export task creates a default output file in your `<disk drive>:\Program Files\HP\HP BTO Software\data\datafiles` directory based on the data type and level of summarization you specified.

scopent Data

When you export scopent log file data, the following default file names are assigned to the exported files.

xfrdGLOBAL.ext	global detail data
xfrsGLOBAL.ext	global summary data
xfrdAPPLICATION.ext	application detail data
xfrsAPPLICATION.ext	application summary data
xfrdPROCESS.ext	process detail data
xfrdDISK.ext	disk device detail data
xfrsDISK.ext	disk device summary data
xfrdCPU.ext	CPU detail data
xfrsCPU.ext	CPU summary data
xfrdFILESYSTEM.ext	filesystem detail data
xfrsFILESYSTEM.ext	filesystem summary data
xfrdNETIF.ext	netif detail data
xfrsNETIF.ext	netif summary data
xfrdTRANSACTION.ext	transaction tracking detail data
xfrsTRANSACTION.ext	transaction tracking summary data
xfrdCONFIGURATION.ext	configuration detail data
xfrdLOGICAL.ext	logical system detail data file
xfrsLOGICAL.ext	logical system summary data file

When the export task completes, you can view the contents of the output export file by clicking the **Examine Data** button in the Export Log File Data dialog box.

The default file names are created from the data type name. The prefix is either **xfrd** or **xfrs** depending on whether the data is detail or summary data. The extension (**.ext**) is the file format specified in the export template file: **asc** (ASCII), **bin** (binary), **dat** (datafile), or **wk1** (spreadsheet).

For example:

xfrdNETIF.wk1 contains detailed data for the NETIF data type in spreadsheet format.

xfrsAPPLICATION.asc contains summarized data for the application data type in ASCII format.

DSI Data

When you export DSI log file data, the default file names are created from the class name. The prefix is either **xfrd** or **xfrs**, depending on whether the data is detail or summary data. The extension is the file format specified in the export template file: **asc** (ASCII), **dat** (datafile), or **wk1** (spreadsheet).

For example:

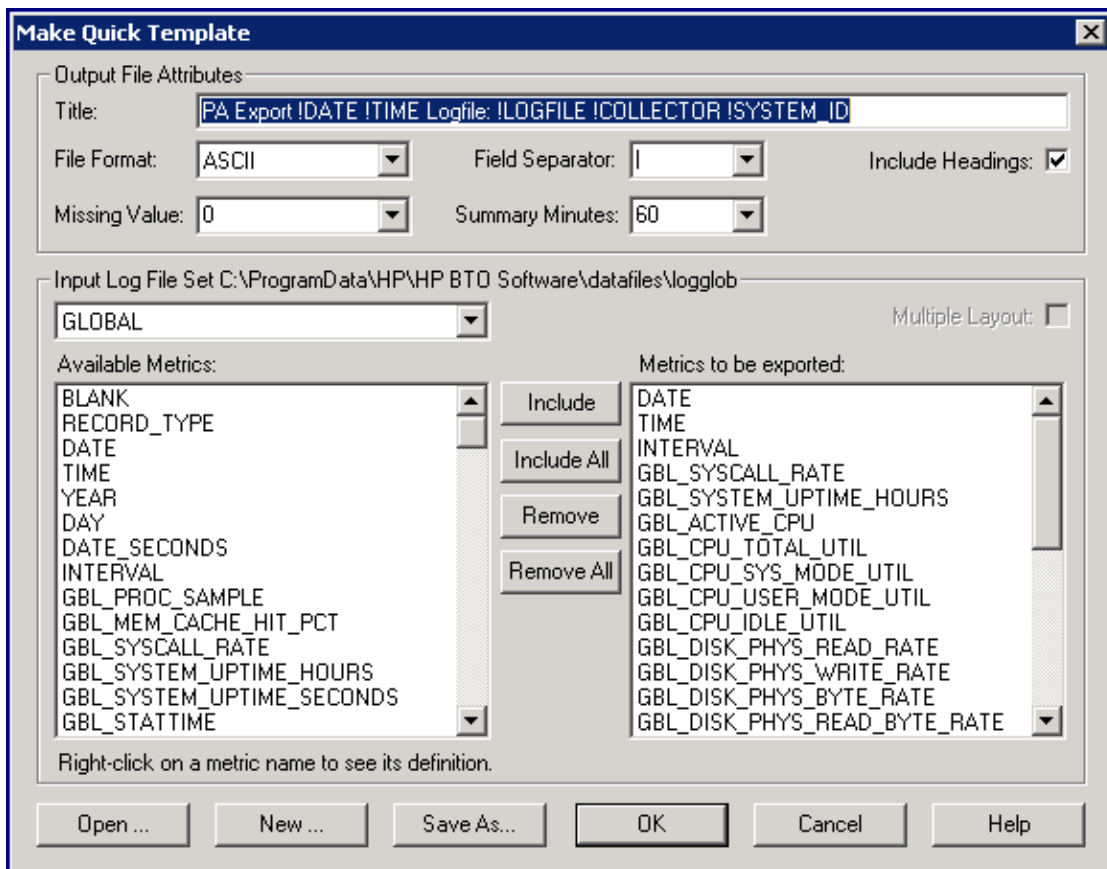
xfrdACCTG.wk1 contains detailed data for the ACCTG class in spreadsheet format.

xfrsPERSONL.asc contains summarized data for the PERSONL class in ASCII format.

Making a Quick Export Template

Use the Make Quick Template function in the Export Log File Data dialog box to select specific metrics to be included in your export file and to change any of the file attributes and metrics that are specified in the export template you selected for your export.

Make Quick Template Dialog Box



To make a quick template, follow these steps:

1. Click the **Make Quick Template** button in the Export Log File Data dialog box. The Make Quick Template dialog box appears, showing the title of the export file selected for the export.
2. The boxes below Output File Attributes show the current settings for the export file, based on the file attributes set in the export template selected for the export. You can modify any of these settings.
3. To use a different export template file, click the **Open** button.
4. You also have the option of clearing all existing settings from the Make Quick Template dialog box and creating a totally new export template file by clicking the **New** button.

Selecting Metrics to Export

After selecting the data type or class of the metrics to be exported, you can select the specific metrics that you want included in the export. Each data type or class has its own set of metrics that are listed under Available Metrics. The metrics listed under "Metrics to be exported" are the metrics specified by the current export template to include in the export. You can either use that list, remove metrics from the list, or select other available metrics to include in the export.

If you select the application, disk, cpu, filesystem, netif, or transaction data types, select the **Multiple Layout** check box to generate multiple layouts (per record output), or leave it cleared to generate single layout.

Saving Your Selections

After making your selections, you can either:

Proceed with the export using the selections you made to the file attributes and to the list of metrics to be exported but WITHOUT saving the changes to any template file.

Save your selections to a new export template file that will be available for use in future exports. The original export template file remains unchanged.

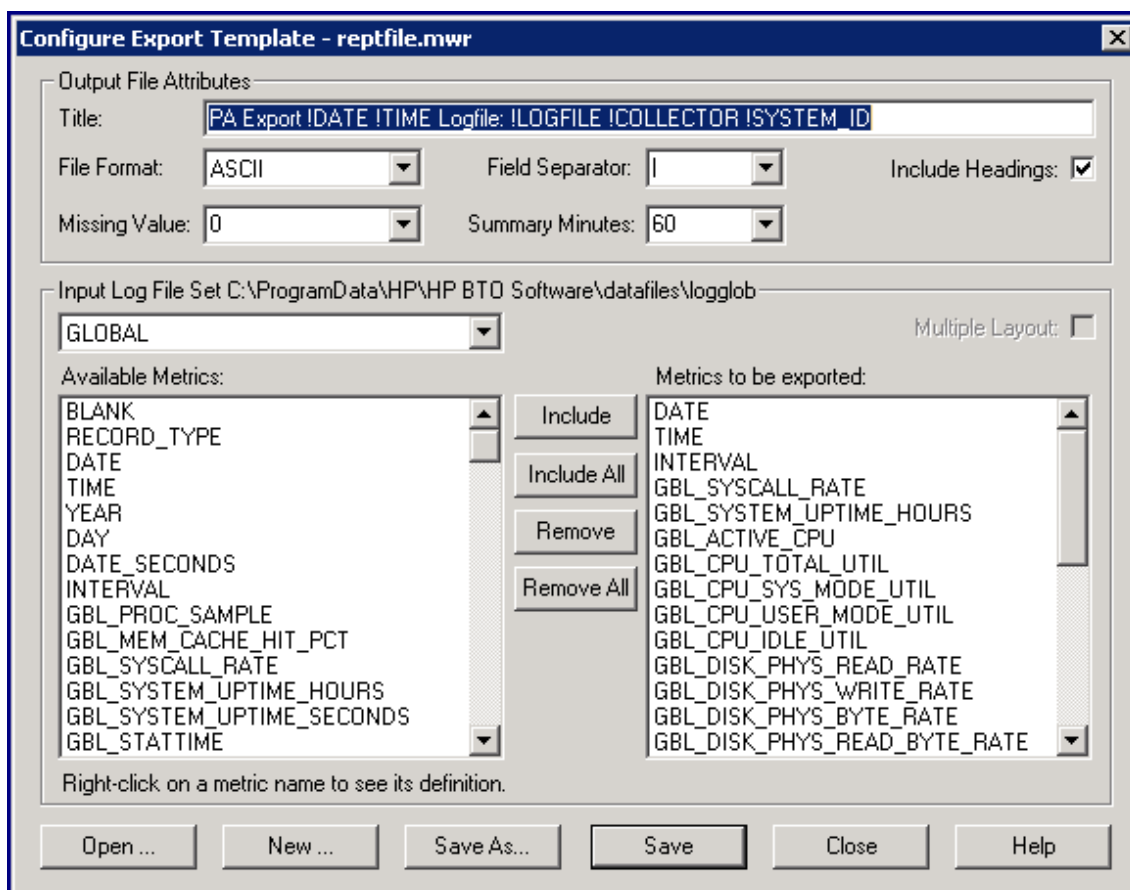
Note: If you save your selections to a new export template file, you must include the **.mwr** file name extension when you specify the new template file name. For example, **mytmplt.mwr**

For step-by-step instructions for making a quick export template, choose **Help Topics** from the Help menu, select "**How Do I...?**," and then select "**Make a quick export template.**"

Configuring Export Templates

Use the **Export Templates** command from the Configure menu to customize an existing export template file or create a new export template file. Use the Configure Export Template dialog box to select new file attributes and specific metrics to be included in the template.

Configure Export Template Dialog Box



To configure an export template, follow these steps:

1. Click **Export Templates** from the Configure menu on the main window. The Configure Export Template dialog box appears showing the name of the currently open export template file in the dialog box title.
2. To edit a different export template file, click the **Open** button.
3. The boxes below Output File Attributes show the current settings for the export file, based on the file attributes set in the export template you are configuring. You can modify any of these settings.
4. You can also clear all existing settings from the Configure Export Template dialog box and create a totally new export template file by clicking the **New** button.

Selecting Metrics for Export

After selecting the data type or class of the metrics to be exported, you can select the specific metrics that you want included in the export. Each data type or class has its own set of metrics that are listed under Available Metrics. The metrics listed under "Metrics to be exported" are the metrics specified by the current export template to include in the export. You can use that list, remove metrics from the list, or select other available metrics to include in the export.

If you select the application, disk, cpu, filesystem, netif, or transaction data types, select the **Multiple Layout** check box to generate multiple layouts (per record output), or leave it cleared to generate single layout.

Saving Your Selections

You have three choices for saving the template you edited:

Save the changes to the current template file.

Save the changes to a new template file, in which case the original template file remains unchanged.

Cancel the changes to avoid making changes to any template file.

If you save your selections to a new export template file, you must include the .mwr file name extension when you specify the new file name.

If you click the **Close** button *after* you make changes to the template file but *before* you saved them, you are prompted to either cancel or save your changes. Clicking the **Cancel** button returns you to the Configure Export Template dialog box.

For step-by-step instructions for configuring an export template file, choose **Help Topics** from the Help menu, select "**How Do I...?**," and then select "**Configure an export template file.**"

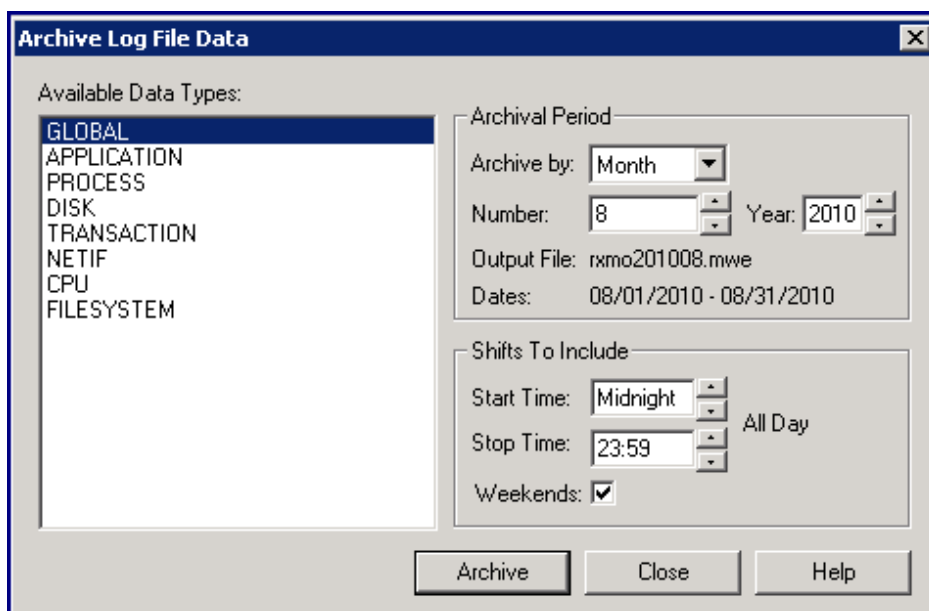
Archiving Log File Data

Use the **Archive** command from the Logfile menu to extract selected portions of scopent log file data for archiving and future data analysis.

For archival purposes, data can only be extracted from the raw log files .

The extracted data is automatically placed in an archival output file in the *<disk drive>*:**Program Files\HP\HP BTO Software\data\datafiles** directory whose name reflects the selected archival period. These files can be copied to tape for offline storage and then deleted to release disk space.

Archive Log File Data Dialog Box



Archival Periods

You can select data to be extracted based on data logged during a specific weekly, monthly, or yearly period.

You can also limit extraction to data that was logged during specific hours of the day that correspond to work shifts and include or exclude weekends (Saturday and Sunday). If no shift is specified, the default is extraction of 24-hours' worth of data for every day. By default, weekends are included.

Appending Archived Data

The archiving function has a special feature. Depending on which archival period you select – weekly, monthly, or yearly – the *previous* output file for that archival period is automatically checked to see if it contains data extracted up to the last day. If not, the data is appended to the file to complete the previous archival period's extraction.

For example, on May 7, 1999, you begin archiving monthly data for May 1999. An output file named **rxmo199905.mwe** is created containing data from May 1 through the current date (May 7).

On June 4, 1999, another monthly archival period is invoked. Before the **rxmo199906.mwe** file is created for June, the **rxmo199905.mwe** file from the previous month is checked. When it is found to be incomplete, data is appended to it to complete the extraction through May 31, 1999. Then, the **rxmo199906.mwe** file is created to hold data from June 1, 1999, to the current date (June 4).

As long as another monthly (weekly, yearly) archival period is invoked at least once a month (week, year), this feature will complete each archival period's file before creating the next archival period's file.

Archiving Tips

Here are some suggestions for archiving your log file data:

Once a month, specify the monthly archival period and extract all the detail data from your raw log files into a single extracted log file.

If your system generates more than 64 MB of data each month, you may need to extract data on a weekly basis, or you can eliminate process detail data from the extraction.

Extract global summary and application summary data on a yearly basis, which should minimize the disk space required. This archive file can then be used for long-term analysis of trends.

To archive log file data, follow these steps:

1. Click **Archive** from the Logfile menu on the main window. The Archive Log File Data dialog box appears. The data to be archived is extracted from the raw log file set.
2. After selecting the type of data to be archived from the Available Data Types list, specify the archival period – Week, Month, or Year, and any shifts to include.
3. Click the **Archive** button to start the archiving process.
4. For step-by-step instructions for archiving log file data, choose **Help Topics** from the Help menu, select "**How Do I...?**," and then select "**Archive log file data.**"

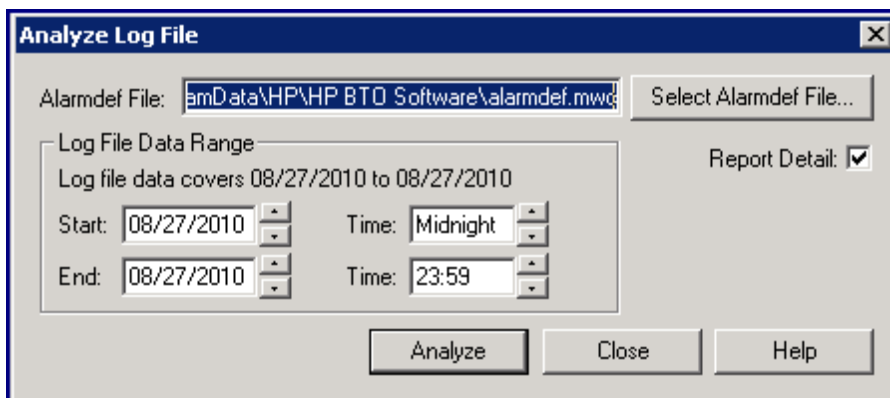
Analyzing a Log File

Use the **Analyze Log File** command from the Logfile menu to analyze data in the raw log file set against alarm definitions in an alarm definitions file, and report on any resulting alarm activity.

This task enables you to evaluate whether or not your alarm definitions are a good match against the historical data collected on your system. It also enables you to decide if your alarm definitions will generate too many or too few alarms on your analysis system.

The raw log files being analyzed are referenced in Performance Collection Component's default data source, SCOPE. To analyze a different log file, place a USE statement in your alarm definitions file that specifies the name of the data source that references that log file.

Analyze Log File Dialog Box



Range of Data to be Analyzed

You can analyze log file data that was collected during a specific period of time. If you do not specify a specific range of data to be analyzed, data is analyzed using the default starting date – 30 days before the latest date in the log file or, if fewer than 30 days of data are present, the date of the earliest record in the log file.

Analysis Report

As this task executes, it generates a printable report that lists alarm events and an alarm summary. (Alarm events are listed only if the Report Detail box in the Analyze Log File dialog box is checked.)

Alarm events include alarm START, END, and REPEAT status plus any text in associated PRINT statements. Also, if any text in PRINT statements are listed as conditions (in IF statements) and become true, the text is included. EXEC statements are not executed but are listed so you can see what would have been executed.

Alarm summaries show a count of the number of alarms that occurred and the amount of time each alarm was active (on). The count includes alarm starts and repeats, but not alarm ends.

To analyze a log file, follow these steps:

1. Click **Analyze** from the Logfile menu in the main window. The **Analyze Log File** dialog box appears showing the name of the currently selected alarm definitions file.
2. To use a different alarm definitions file, click the **Select Alarmdef** File button.
3. Select the range of log file data to be analyzed.
4. Check the **Report Detail** box if you want to include alarm events in the analysis report. Otherwise, only the alarm summary is generated.
5. Click the **Analyze** button to start the analysis. The analysis results are displayed in a MeasureWare Agent Report Viewer window.

For step-by-step instructions for analyzing a log file, choose **Help Topics** from the Help menu, select "**How Do I...?**," and then select "**Analyze a log file.**"

Scanning a Log File

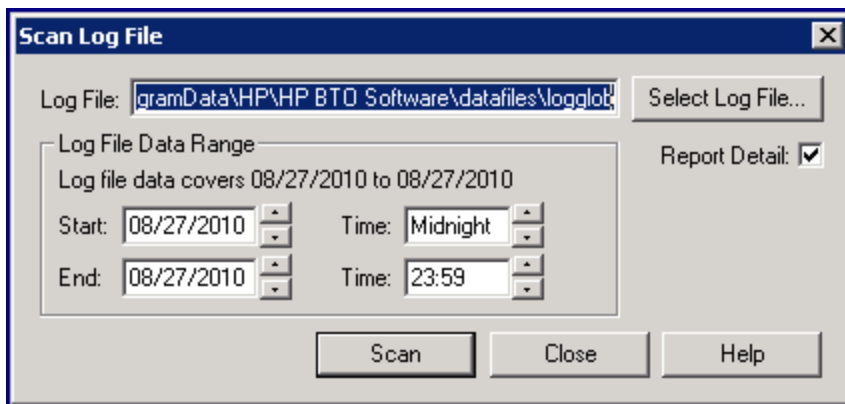
Use the **Scan Log File** command from the Logfile menu to scan a scopent log file and create a report on its contents. You can either scan an entire log file or scan portions of a log file for data that was collected during a specific period of time.

The report produced by the scan consists of 12 sections. The following four sections of the report are always printed.

- Process summary report
- Collector coverage summary
- Log file contents summary
- Log file empty space summary

The following eight sections of the report are printed only if you select **Report Detail** in the Scan Log File dialog box.

- Initial **parm** file global information and system configuration information
- Initial **parm** file application definitions
- **parm** file global changes
- **parm** file application/change notifications
- Collector off-time notifications
- Application-specific summary reports



To scan a log file, follow these steps:

1. Click **Scan Log File** from the Logfile menu on the main window. The Scan Log File dialog box appears with the name of the currently open log file highlighted.
2. To scan a different log file, click the **Select Log File** button.
3. To scan data that was logged during a specific time period, under Log File Data Range, select or type the dates and times for the beginning and end of that time period.
4. Check the **Report Detail** box if you want a complete scan report. Otherwise, only a subset of the report is generated.

5. To start the scan process, click the **Scan** button. The scan results are shown in a Performance Collection Component Report Viewer window.
6. For step-by-step instructions for scanning a log file, choose **Help Topics** from the Help menu, select "**How Do I...?**," and then select "**Scan a log file.**"

Resizing a Log File

Use the **Resize Log File** command from the Logfile menu to change the size of your raw scopent log files. You can resize using either the size in megabytes for the given file or the number of days of data the file should hold.

The maximum size of a raw log file is specified in the size parameter in the **parm** file. Resizing the log file gives you more control over how often the log file data is rolled back.

You can select any of the following types of data to resize: global, application, process, device, or transaction, which correspond to the raw log files **logglob**, **logappl**, **logproc**, **logdev**, and **logtran**. You then choose how the resize will be performed – in megabytes or by number of days. Depending on which type of resize you choose, the Log File Settings box in the Resize Log File dialog box shows the following:

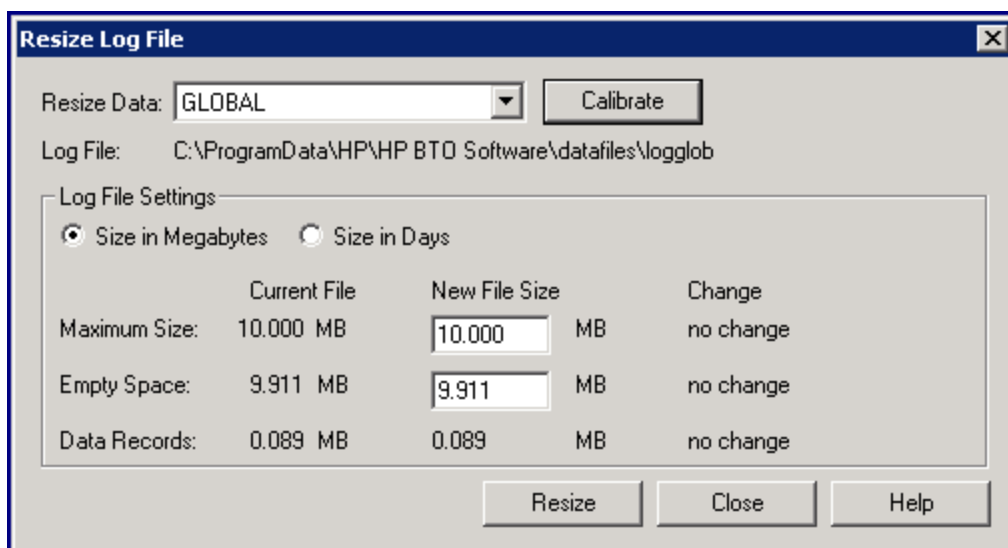
The Maximum Size fields show current file size, the new file size, and the change made by the resize.

The Empty Space fields show the amount of room in the current file, the amount required in the file after the resizing process is complete, and the change. These values are used to determine if any of the data currently in the log file must be removed in the resizing process.

The Data Records fields show the amount of data records contained in the current log file and the new amount of data records that will be in the resized log file.

Log file sizes are maintained in megabytes. Often it is more convenient to specify sizes in days rather than megabytes. If you select "Size in Days", all units on the dialog box will change to "days". The conversion from megabytes to days is based on a "megabytes-per-day" value for each type of data. Initially, estimated values are used for this conversion.

A more precise value can be obtained by clicking the **Calibrate** button. The calibrate function actually measures the existing log files for more precise megabytes-per-day values. If you are specifying size in megabytes, no conversion is needed and the calibrate function need not be used.



Before resizing a log file, you *must* stop the scopent collector. To stop scopent, follow the steps in [Stopping and Restarting Data Collection](#).

Attempting to resize a log file without first stopping scopent will not affect the existing log file. To resize a log file, follow these steps:

- Once scopent is stopped, choose **Resize Log File** from the Logfile menu on the main window to display the Resize Log File dialog box.
- In the Resize Data box, select the type of data to be resized: **global**, **application**, **process**, **device**, or **transaction**.
- Select either **Size in Megabytes** or **Size in Days**. Depending on what you selected, the Current File and New File Sizes are shown.
- To perform the resize based on the New File Sizes shown, click the **Resize** button to start the resize process.
- To get a more accurate estimate of how much additional space to add to the log file when sizing in days, follow these steps:
 - Click the **Calibrate** button. Within moments, the actual number and size of the data records that were logged in the file during the last 30 days are displayed.
 - Click the **Close** button to return to the Resize Log File dialog box. New Current File and New File Size values based on the calibration are shown, if sizing in days.
 - Click the **Resize** button to resize the log file.

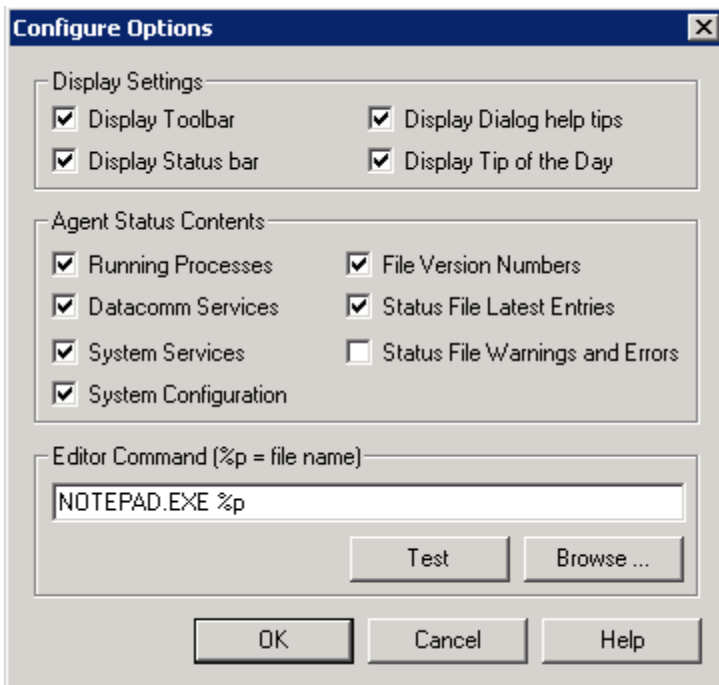
Before performing another task, start scopent using the steps in [Stopping and Restarting Data Collection](#).

For step-by-step instructions for resizing a log file, choose **Help Topics** from the Help menu, select "**How Do I...?**," and then select "**Resize a log file.**"

Configuring User Options

Use the **Options** command from the Configure menu to control the display of the toolbar, status bar, dialog help tips, and Tip of the Day on your main window and while you are using Performance Collection Component.

You can also use the **Options** command to configure an editor or word processor for modifying the collection parameters and alarm definitions files and to select the type of status information you want to view when you choose the **Status** command from the Agent menu.



To configure user options, follow these steps:

1. Click the **Options** command from the Configure menu in the main window to display the Configure Options dialog box.
2. Select the **Display Toolbar** check box to display the toolbar in the main window.
3. Select the **Display Status Bar** check box to display current status at the bottom of each dialog box and in the main window.
4. Select the **Display Dialog Help Tips** check box to display help tips within dialog boxes.
5. Select the Display Tip of the Day check box to display the current day's tip when you open the Performance Collection Component main window.

To configure, follow these steps:

1. Type the editor's directory path and file name in the Editor Command box, using the `.exe` file name extension (for example, `C:\MSOffice\winword\winword.exe`).

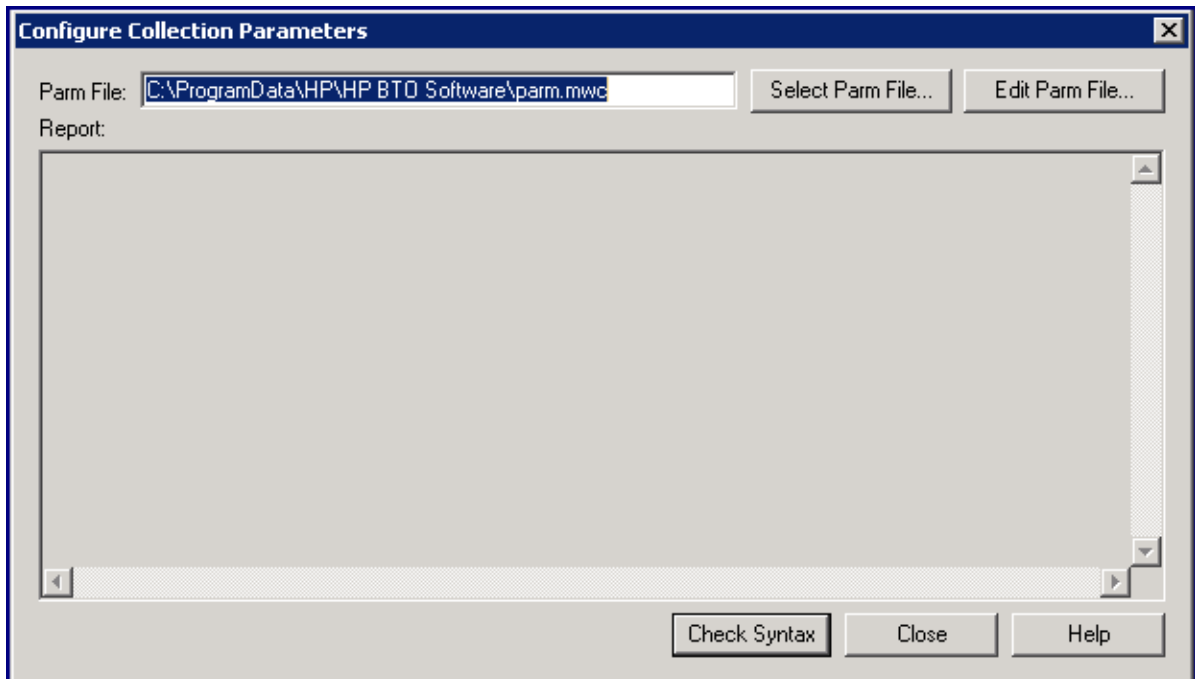
2. Click the **Browse** button to display the Select a Text Editor dialog box from which you can select your editor.
3. Click the **Test** button to make sure that the editor you selected is configured and then click **OK**.
4. To configure which agent status information you want to view, select one or more of the option boxes shown under Agent Status Contents, and then click **OK**.
5. For step-by-step instructions for configuring user options, choose **Help Topics** from the Help menu, select "**How Do I...?**," and then select "**Configure user options.**"

Configuring Collection Parameters

Use the **Collection Parameters** command from the Configure menu to check the syntax of the **parm** file that is used by scopent for data collection. You can examine the **parm** file's settings for syntax errors and warnings and to see how much room is available for defining applications.

If any warnings or errors are found and you want to correct them, or if you want to change or add **parm** file parameters, you can easily modify the **parm** file using the Edit Parm File function.

A detailed description of the **parm** file and its parameters can be found in [Managing Data Collection](#).



To check the syntax, follow these steps:

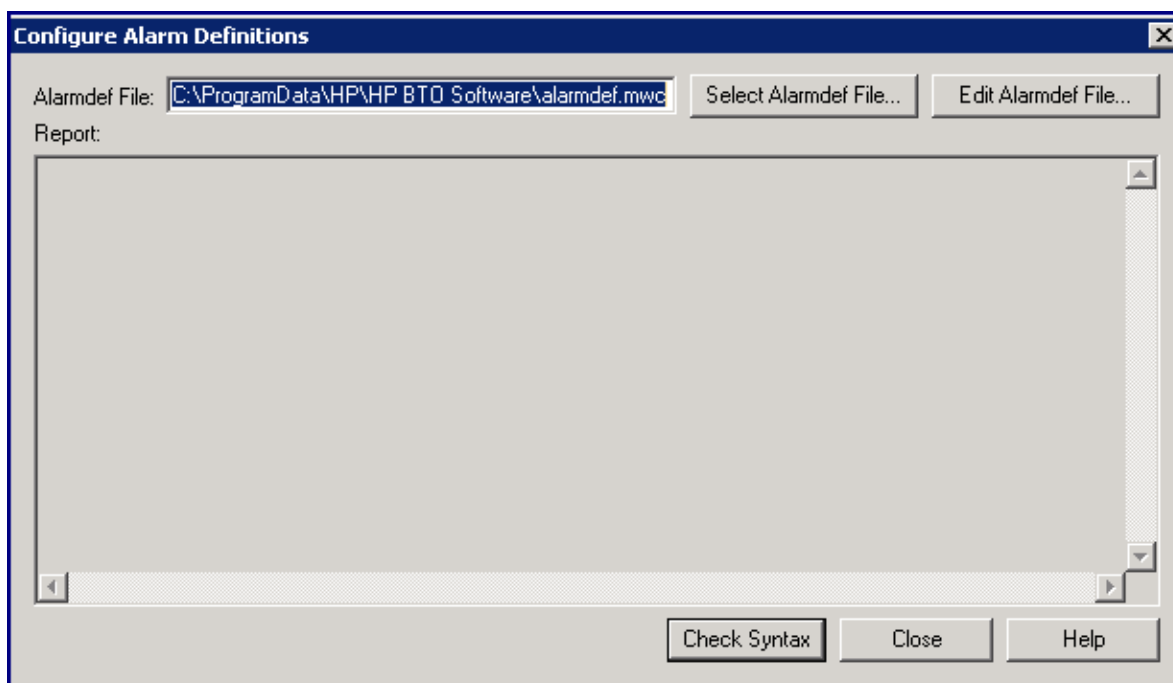
1. Click **Collection Parameters** from the Configuration menu on the Performance Collection Component main window. The Configure Collection Parameters dialog box appears showing the name of the currently open **parm.mwc** file in the Parm File box.
2. To check a different **parm** file, click the **Select Parm File** button.
3. To check the syntax of the **parm** file, click the **Check Syntax** button. Any resulting warnings or errors are displayed in the Performance Collection Component Report Viewer window.

4. To modify any portion of the **parm** file, click the **Edit Parm File** button. You can position the Edit Parm File and the Configure Collection Parameters dialog boxes on your screen so that you can use both at the same time.
5. For step-by-step instructions for checking the syntax of the **parm** file, choose **Help Topics** from the Help menu, select "**How Do I...?**," and then select "**Check the syntax of a collection parameters file.**"

Configuring Alarm Definitions

You use the **Alarm Definitions** command from the Configure menu to check the syntax of the alarm definitions in an alarm definitions file (**alarmdef.mwc**). When you determine that the alarm definitions syntax is correct, you can analyze a log file against the alarm definitions to check for alarms in historical log file data (see [Analyzing a Log File](#)).

If any warnings or errors are found and you want to correct them, or if you want to add or delete alarm definitions, you can easily modify the alarm definitions file using the **Edit Alarmdef File** button in the Configure Alarm Definitions dialog box.



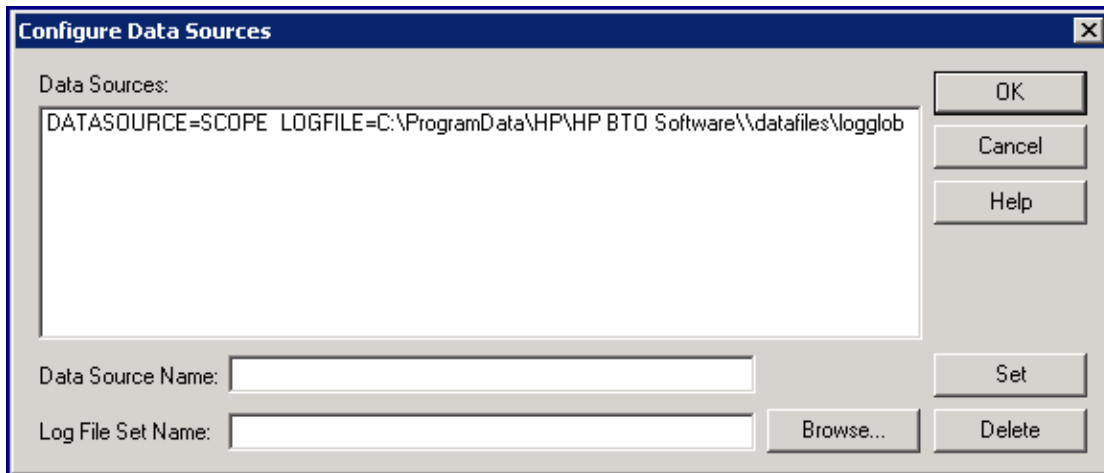
To check the syntax, follow these steps:

1. Click **Alarm Definitions** from the Configure menu on the Performance Collection Component main window. The Configure Alarm Definitions dialog box appears showing the name of the currently open alarm definitions file.
2. To check a different alarm definitions file, click the **Select Alarmdef File** button.
3. Click the **Check Syntax** button to start the syntax checking process. After a few seconds, the checking results are displayed, including any warnings or errors, in the Performance Collection Component Report Viewer window.

4. To modify any portion of the alarm definitions file, click the **Edit Alarmdef File** button.
5. For step-by-step instructions for checking the syntax of an alarm definitions file, choose **Help Topics** from the Help menu, select "**How Do I...?**," and then select "**Checking the syntax of an alarm definitions file.**"

Configuring Data Sources

The Performance Collection Component uses data sources for each specific data source such as scopent log files or DSI log files. Each data source consists of a single log file set. The data source is configured in the **datasources** file that resides in the **<DataDir>\conf\perf** directory. When you first start up Performance Collection Component after installation, a default data source named SCOPE is already configured and provides a scopent log file set.



Data Sources File Format

Each entry you place into the **datasources** file represents a data source consisting of one log file set. The entry specifies the data source name by which the repository server is to be known and where the data it contains is to be found. Entries are case-insensitive. The syntax is:

datasource=*datasource_name* **logfile=***logfile_set*

datasource is a keyword. *datasource_name* is the name used to identify the data source used in alarm definitions or analysis software. Data source names must be unique. The maximum length for the *datasource_name* is 64 characters.

logfile is a keyword. *logfile_set* is the fully qualified name that identifies the log file set. It can be a raw log file set created by scopent, an extracted log file created by the **extract** task, or a DSI log file set. If you specify a log file path name that contains embedded blanks, you must place double quotes (") around the path name.

When specifying a scopent log file set, use only the **logglob** file name. You do not need to specify other raw log file names because they are accessed as a single log file set.

The same applies when specifying a DSI log file set. Specify only the name of the DSI root file. You do not need to specify any of the other files in the DSI log file set.

Configuring Data Sources from Remote Locations

The universal naming convention (UNC) is required when you specify a log file set that resides on a network share. At system start-up, the Performance Collection Component service is started automatically, and drive mappings for remotely connected file systems are not established until the user logs on. Therefore, any data source that uses a drive-mapped name to reference a log file on a remote system causes Coda to generate an invalid data source error. If you start the Performance Collection Component service *after* logging on, the data source is processed because the drive mappings are now established.

Here are three examples of data source entries:

Example 1:

The following example shows the default SCOPE data source residing on the default

```
<disk drive>:\Program Files\HP\HP BTO Software\data\datafiles\directory  
datasource=SCOPE logfile="C:\Program Files\HP\HP BTO  
Software\data\datafiles\logglob"
```

Example 2:

In the following example, the universal naming convention (UNC) is used to specify a log file set that resides on a network share.

```
datasource=RXLOG logfile=\\lab_sys\my_share\rxlog
```

Example 3:

The following example shows the SCOPE data source residing in a directory whose path name contains an embedded blank.

```
datasource=SCOPE logfile="C:\Program Files\HP\HP BTO Software\data\donna test\logglob"
```

To configure data sources, Click **Data Sources** from the Configure menu on the Performance Collection Component main window.

The Configure Data Sources dialog box appears listing the current data source entries. Each entry represents a single data source.

To modify, follow these steps:

1. Select the data source in the Data Sources list.
2. Click the **Log File Set Name** box, modify the log file set name, and click the **Set** button.

To add a new data source, follow these steps:

1. Click the **Data Source Name** box and enter a new name.
2. Click the **Log File Set Name** box, enter a new fully qualified log file set name, and click the **Set** button.

Or,

3. Click the **Browse** button to select an existing data source.

To delete a data source, follow these steps:

1. Select the data source in the Data Sources list.
2. Click the **Delete** button.
3. When you finish configuring your data sources file, click **OK**.

Activate the changes:

Before proceeding with another task, you must activate any changes you made to the data sources. Perform the following steps:

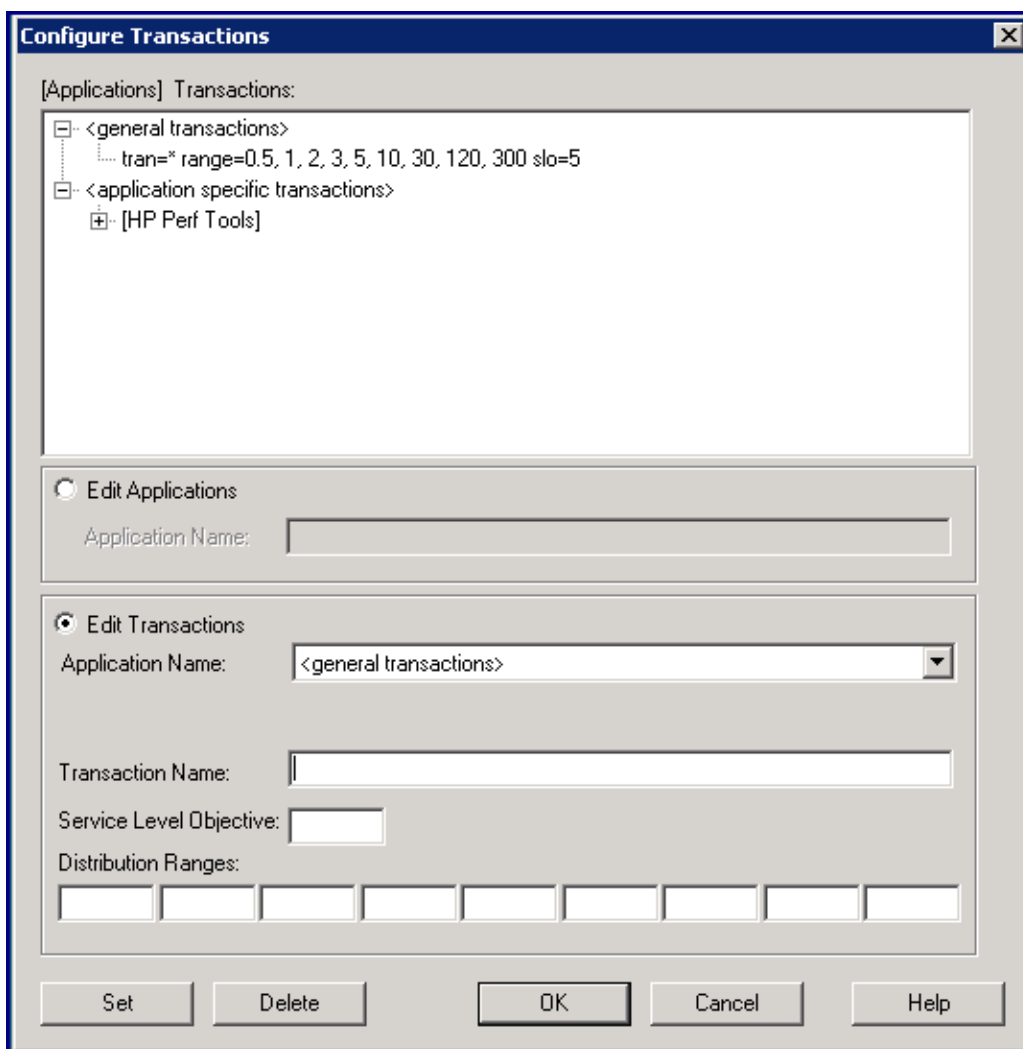
1. Choose **Start/Stop** from the Agent menu on the Performance Collection Component main window to open the MeasureWare Services window.
2. Click the **Stop Services** button to stop MeasureWare services.
 - When the **Stop Services** button appears dimmed, click the **Start Services** button.
4. Click the **Close** button to return to the main window.

For step-by-step instructions for modifying the data sources file, choose **Help Topics** from the Help menu, select "**How Do I...?**," and then select "**Modify a data source file.**"

Configuring Transactions

You use the transaction configuration file, **ttdconf.mwc**, to customize collection of transaction data for an application. The file defines the transaction name, performance distribution range, and the service level objective you want to meet for each transaction. Optionally, you can define transactions that are specific to an application.

The default **ttdconf.mwc** file contains three entries. Two entries define transactions used by the Performance Collection Component scoped collector, and a third entry, **tran=*** registers all transactions in applications that were instrumented with Application Response Measurement (ARM) API function calls.



If you are adding new applications to your system that use the service level objective and range values from the `tran=*` entry in the default `ttdconf.mwc` file, you do not have to do anything to incorporate the new transactions. All of the default values are applied automatically to them.

However, if you are adding applications to your system that have transactions with their own unique service level objectives and distribution range values, you must add these transactions to the `ttdconf.mwc` file.

Note: The order of the entries in the `ttdconf.mwc` file is not relevant. Exact matches are sought first. If none are found, the longest match with a trailing asterisk (*) is used.

Before you make any changes to the file, see [What is Transaction Tracking?](#) for descriptions of the configuration file format, transaction and application names, performance distribution ranges, and service level objectives. To configure, click **Transactions** from the Configure menu on the Performance Collection Component main window to display the Configure Transactions dialog box. Using this dialog box, you can perform the following tasks:

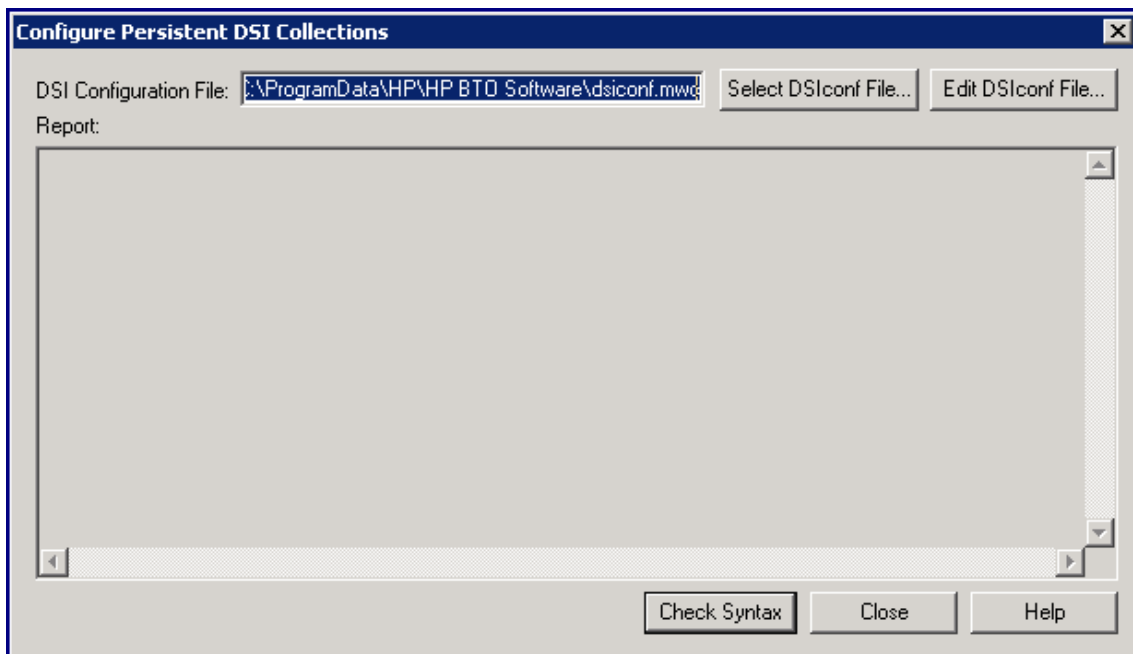
- Add a general transaction
- Add an application-specific transaction

- Modify a transaction's performance distribution range or service level objective
- Delete a transaction

For step-by-step instructions for performing these tasks, choose **Help Topics** from the Help menu, select "**How Do I...?**," and then select "**Configure transactions.**"

Configuring Persistent DSI Collections

Use the **Persistent DSI Collections** command from the Configure menu to check the syntax or modify the DSI configuration file, **dsiconf.mwc**. The **dsiconf.mwc** file is used to configure continuous logging of data collections that were brought into Performance Collection Component from outside sources. For more information, see [Overview of Data Source](#).



To check the syntax of the DSI configuration file, follow these steps:

1. Click **Persistent DSI Collections** from the Configure menu on the Performance Collection Component main window. The Configure Persistent DSI Collections dialog box shows the name of the currently open **dsiconf.mwc** file.
2. To check a different **dsiconf.mwc** file, click the **Select DSIconf File** button.
3. To check the syntax of the file, click the **Check Syntax** button. Any resulting warnings or errors are displayed in the Performance Collection Component Report Viewer window.
4. To modify any portion of the file, click the **Edit DSIconf File** button. You can position the Edit DSIconf File and the Configure Persistent DSI Collections dialog boxes on your screen so that you can use both at the same time.

For step-by-step instructions for checking the syntax of the DSI configuration file, choose **Help Topics** from the Help menu, select "**How Do I...?**," and then select "**Check the syntax of a DSI configuration file.**"

To modify a DSI configuration file, follow these steps:

1. Click **Persistent DSI Collections** from the Configure menu on the Performance Collection Component main window and then click the **Edit DSIconf File** button in the Configure Persistent DSI Collections dialog box. The contents of the currently open **dsiconf.mwc** file are displayed in a previously specified editor or word processor. (To specify an editor or word processor, see [Configuring User Options](#))
2. Before you make any changes to the file, see [Using the Performance Collection Component on Windows](#) for rules and conventions to follow.
3. Modify the file as necessary and save the file in text format.

Before proceeding with another task, you *must* activate any changes you made to the **dsiconf.mwc** file. Perform the following steps:

1. Click **Start/Stop** from the Agent menu on the Performance Collection Component main window to open the MeasureWare Services window.
2. Select the **Persistent DSI Collections** check box.
3. Click the **Refresh** button.
4. Click the **Close** button to return to the main window.

For step-by-step instructions for modifying a DSI configuration file, choose **Help Topics** from the Help menu, select "**How Do I...?**," and then select "**Modify a DSI configuration file.**"

Note: If you use WordPad, Notepad, or Microsoft Word to modify your **dsiconf.mwc** file and then use the **Save As** command to save the file, the **default .txt** extension will automatically be added to the file name. You will then have a file named **dsiconf.mwc.txt**. To retain the **dsiconf.mwc** file name, use the **Save As** command to save your file as a text file and enclose the file name in double quotes ("). For example: "**dsiconf.mwc**"

Checking Performance Collection Component Status

Use the **Status command** from the Agent menu to review the current status of Performance Collection Component processes. The information is generated by the **perfstat** program.

You can designate which specific information to include in the status report by choosing the **Options** command from the Configure menu display and selecting any of the following options in the Configure Options dialog box.

Running Processes

Background and foreground processes that are currently running for Performance Collection Component are listed. Any background processes that should be running but are *not* running are listed.

Datacomm Services

Datacomm services locate and communicate with the Performance Collection Component datacomm services. They show whether or not the alarm generator database server (agdbserver) process is running and responsive. If data communications are not enabled, this information may take more than 30 seconds to generate while it waits for datacomm services to respond.

System Services

The current status of Performance Collection Component System Services such as the Scope Collector, Transaction Manager, and Measurement Interface is shown.

System Configuration

System name, operating system version, and processor type.

File Version Numbers

Version numbers of Performance Collection Component files. Any critical files that are missing are noted.

Status File Latest Entries

The latest few entries from each performance tool status file.

Status File Warnings and Errors

Any lines from the performance tool status files that contain "Error" or "Warning" are listed. A very large listing can be produced in cases where warnings have been ignored for long periods of time.

To list the current status, click **Status** from the Agent menu on the Performance Collection Component main window. The Performance Collection Component Report Viewer displays the information you selected from the Configure Options dialog box.

To get a complete report of all status information, click **Report** from the Agent menu. The Performance Collection Component Report Viewer displays a complete list of all status information.

For step-by-step instructions for checking Performance Collection Component status, choose **Help Topics** from the Help menu, select "**How Do I...?**," and then select "**Check status of Performance Collection Component processes.**"

You can also run the **perfstat** program from the Windows Command Prompt.

Building Collections of Performance Counters

Performance Collection Component provides access to Windows performance counters that are used to measure system, application, or device performance on your system. You use the Extended Collection Builder and Manager (ECBM) to select specific performance counters to build data collections.

Building a Performance Counter Collection

To build a collection, choose **Extended Collections** from the Agent menu on the Performance Collection Component main window. The Extended Collection Builder and Manager window appears, showing a list of Windows objects in the left pane. For instructions on building collections, choose **Help Topics** from the Help menu in the Extended Collection Builder and Manager window.

After you build your collections of Windows performance counters, use the Extended Collection Manager pane at the bottom to register, start, and stop new and existing collections.

Managing a Performance Counter Collection

To manage your data collections, use the Extended Collection Manager pane at the bottom of the Extended Collection Builder and Manager. Initially, no collections appear because you must register a collection before you can start collecting data.

After you register or store the collection you created, the Extended Collection Manager pane shows a list of current collections. The Extended Collection Manager pane also displays the state of every collection and enables you to view information (properties) about the collection itself. For instructions on managing your collections, choose **Help Topics** from the **Help** menu in the Extended Collection Builder and Manager window.

Tips for Using Extended Collection Builder and Manager

The `<InstallDir>\paperdocs\mwa\C\monxref.txt` file contains a cross-reference of Performance Collection Component metrics to Windows performance counters and commands. Logging data through the Extended Collection Builder and Manager for metrics already collected by Performance Collection Component incurs additional system overhead.

When you use the Extended Collection Builder to create collections, default metric names are assigned to Windows performance counters for internal use with the Performance Collection Component. These default names are generally not meaningful or easy to decipher. To make metric names more meaningful or match them to the metric names supplied by their source application, modify metric attributes by right-clicking or double clicking the metric name after you drag it from left to right pane in the Extended Collection Builder and Manager window. (See the Extended Collection Builder and Manager online help for detailed instructions.)

If you start 61 or more collections, the collections beyond 60 go into error states. This may cause problems with other collections.

If you collect logical disk metrics from a system configured with **Wolfpack**, you must restart the collection in order to collect data for any new disk instances not present when the collection was registered.

Successful deletion of collections requires restarting Performance Collection Component after deleting the collection. If Performance Collection Component is not restarted, you might get an error during the delete operation. This error typically means that some files were not successfully deleted. You may need to manually delete any files and directories that remain after you restart Performance Collection Component.

Extended Collection Builder and Manager may report missing values for some metrics with cache counters. The problem may occur under some circumstances when a metric value gets an overflow. A message is also sent to the ECBM status file. You can resolve the problem by restarting the collection.

Explanations of Extended Collection Builder and Manager concepts, and instructions on creating and viewing data collections are available in the Extended Collection Builder and Manager online help. To view online help, from your desktop select **Start® Programs® HP ® Operations Agent® Performance Collection Component® ECB-ECM Online Help**. You can select **Extended Collections** from the Agent menu in the Performance Collection Component main window and select **Help Topics** from the Help menu in the Extended Collection Builder and Manager window. Online help is available by selecting the **Help** button in the dialog boxes that appear in the Extended Collection Builder and Manager.

Administering ECBM from the Command line

You can run the ECBM program from the <rpmtools>\bin directory using the Windows Command prompt.

- Collections can be managed from the command line using the following command:

```
\rpmtools\bin\mwcmcmd.exe
```

- To display various options, type the following command:

```
\rpmtools\bin\mwcmcmd /?
```

- To start the stopped collections, type the following command:

```
mwcmcmd start <collection_name(s)>
```

- To start a new collection from a variable-instance policy, type the following command:

```
mwcmcmd start <policy_name> <collect_name> <instance(s)> [options]
```

The following options are available:

-i <sampling_interval>- change sampling interval (seconds)

-l <logfile_path_name> - change default log location

-a <alarm_file> - change the alarm definitions file

- To stop the active collections, type the following command:

```
mwcmcmd stop <collection_name(s)>
```

- To register a policy file, type the following command:

```
mwcmcmd register <policy_file> <collection/policy_name> [options]
```

The following options are available only when registering a fixed-instance policy file:

-i <sampling_interval> - change sampling interval (seconds)
-l <logfile_path_name> - change default log location
-a <alarm_file> - change the alarm definitions file

- To delete a single collection:

`mwcmcmd delete <collection/policy_name> [options]`

The following options are only available when deleting a collection:

-p <archive_path> - archives logfiles to specified path
-r - restarts Performance agent

- To delete multiple collections or policies:

`mwcmcmd delete { <collection/policy_name(s)> | -c | -all }`

-c - deletes ALL collections
-a - deletes ALL collections and policies

Note: When deleting more than one policy/collection at a time, Performance Collection Component will be automatically restarted, and all associated logfiles will be deleted.

- To list all registered collections and policies, type the following command:

`mwcmcmd list`

- To list the properties of a collection or policy, type the following command:

`mwcmcmd properties <collection/policy_name>`

- To list the variable-instance objects in a policy, type:

`mwcmcmd objects <policy_name>`

Chapter 13: Overview of Data Source Integration

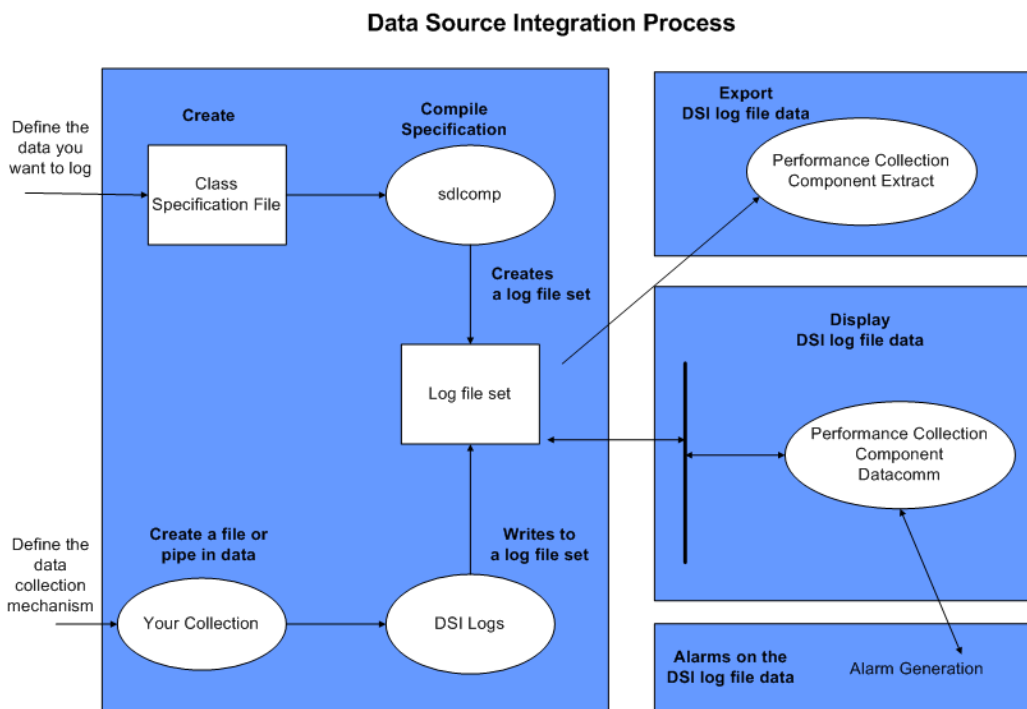
The Data Source Integration (DSI) technology allows you to use the HP Operations agent to log data, define alarms, and access metrics from new sources of data beyond the metrics logged by the Performance Collection Component's scope collector. Metrics can be acquired from data sources such as databases, LAN monitors, and end-user applications.

The data you log using DSI can be displayed in HP Performance Manager along with the standard performance metrics logged by the scope collector. DSI logged data can also be exported, using the Performance Collection Component **extract** program, for display in spreadsheets or similar analysis packages.

How DSI Works

The following diagram shows how DSI log files are created and used to log and manage data. DSI log files contain self-describing data that is collected outside of the Performance Collection Component scope collector. DSI processes are described in more detail on the next page.

Figure 17 Data Source Integration Process



Using DSI to log data consists of the following tasks:

Creating the Class Specification

You first create and compile a specification for each class of data you want to log. The specification describes the class of data as well as the individual metrics to be logged within the class. When you compile the specification using the DSI compiler, **sdlcomp**, a set of empty log files are created to accept data from the **dsilog** program. This process creates the log file set that contains a root file, a description file, and one or more data files.

Collecting and Logging the Data

Then you collect the data to be logged by starting up the process of interest. You can either pipe the output of the collection process to the **dsilog** program directly or from a file where the data was stored. **dsilog** processes the data according to the specification and writes it to the appropriate log file. **dsilog** allows you to specify the form and format of the incoming data.

The data that you feed into the DSI process should contain multiple data records. A record consists of the metric values contained in a single line. If you send data to DSI one record at a time, stop the process, and then send another record, **dsilog** can append but cannot summarize the data.

Using the Data

You can use Performance Manager to display DSI log file data. Or you can use the Performance Collection Component **extract** program to export the data for use with other analysis tools. You can also configure alarms to occur when DSI metrics exceed defined conditions.

Chapter 14: Using Data Source Integration

This chapter is an overview of how you use DSI and contains the following information:

- Planning data collection
- Defining the log file format in the class specification file
- Creating the empty log file set
- Logging data to the log file set
- Using the logged data

For detailed reference information on DSI class specifications and DSI programs, see [DSI Class Specification Reference](#) and [DSI Program Reference](#).

Planning Data Collection

Before creating the DSI class specification files and starting the logging process, you need to address the following topics:

- Understand your environment well enough to know what kinds of data would be useful in managing your computing resources.
- What data is available?
- Where is the data?
- How can you collect the data?
- What are the delimiters between data items? For proper processing by **dsilog**, metric values in the input stream must be separated by blanks (the default) or a user-defined delimiter.
- What is the frequency of collection
- How much space is required to maintain logs?
- What is the output of the program or process that you use to access the data?
- Which alarms do you want generated and under what conditions?
- What options do you have for logging with the class specification and the **dsilog** process?

Defining the Log File Format

Once you have a clear understanding of what kind of data you want to collect, create a class specification to define the data to be logged and to define the log file set that will contain the logged data. You enter the following information in the class specification:

- Data class name and ID number
- Label name (optional) that is a substitute for the class name. (For example, if a label name is present, it can be used in Performance Manager.)

- What you want to happen when old data is rolled out to make room for new data. For more information about how log files are organized, see [How Log Files are Organized](#).
- Metric names and other descriptive information, such as how many decimals to allow for metric values.
- How you want the data summarized if you want to log a limited number of records per hour.

Here is an example of a class specification:

```
CLASS VMSTAT_STATS = 10001
```

```
LABEL "VMSTAT data"
```

```
INDEX BY HOUR
```

```
MAX INDEXES 12
```

```
ROLL BY HOUR
```

```
RECORDS PER HOUR 120;
```

```
METRICS
```

```
RUN_Q_PROCS = 106
```

```
LABEL "Procs in run q"
```

```
PRECISION 0;
```

```
BLOCKED_PROCS = 107
```

```
LABEL "Blocked Processes"
```

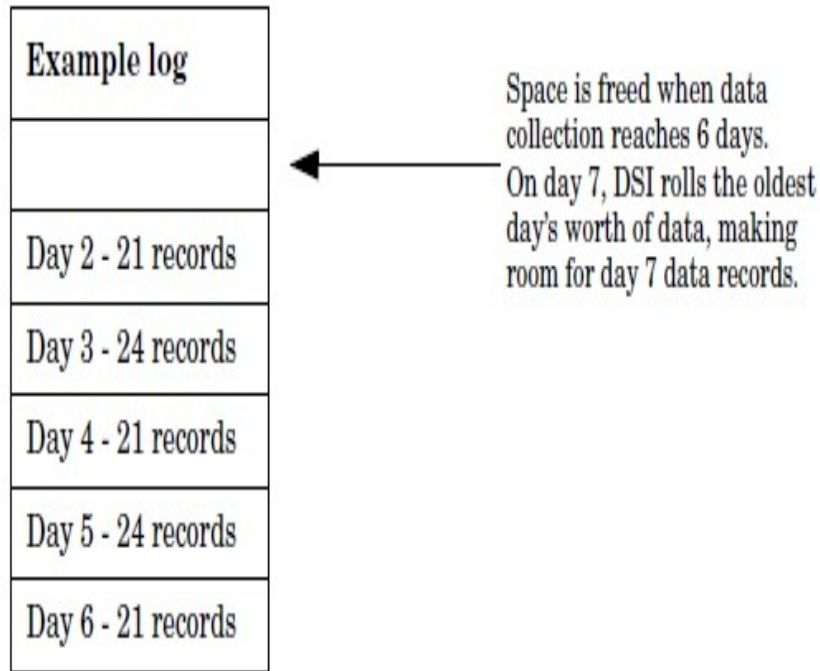
```
PRECISION 0;
```

You can include one class or multiple classes in a class specification file. When you have completed the class specification file, name the file and then save it. When you run the DSI compiler, `sdcomp`, you use this file to create the log file set. For more information about class specification and metric description syntax, see [DSI Class Specification Reference](#).

How Log Files Are Organized

Log files are organized into classes. Each class, which represents one source of incoming data, consists of a group of data items, or metrics, that are logged together. Each record, or row, of data in a class represents one sample of the values for that group of metrics.

The data for classes is stored on disk in log files that are part of the log file set. The log file set contains a root file, a description file, and one or more log files. All the data from a class is always kept in a single data file. However, when you provide a log file set name to the `sdcomp` compiler, you can store multiple classes together in a single log file set or in separate log file sets. The figure below illustrates how two classes can be stored in a single log file set.



Because each class is created as a circular log file, you can set the storage capacity for each class separately, even if you have specified that multiple classes should be stored in a single log file set. When the storage capacity is reached, the class is “rolled”, which means the oldest records in the class are deleted to make room for new data.

You can specify actions, such as exporting the old data to an archive file, to be performed whenever the class is rolled.

Creating the Log File Set

The DSI compiler, `sdlcomp`, uses the class specification file to create or update an empty log file set. The log file set is then used to receive logged data from the `dsilog` program.

To create a log file set, complete the following tasks:

1. Run `sdlcomp` with the appropriate variables and options. For example,
`sdlcomp [-maxclass value] specification_file [logfile_set[log file]] [options]`
2. Check the output for errors and make changes as needed.
For more information about `sdlcomp`, see the [Compiler Syntax](#).

Testing the Class Specification File and the Logging Process (Optional)

DSI uses a program, **sdlgendata**, that allows you to test your class specification file against an incoming source of generated data. You can then examine the output of this process to verify that DSI can log the data according to your specifications. For more information about **sdlgendata**, see [Testing the Logging Process with Sdlgendata](#).

To test your class specification file for the logging process:

1. Feed the data that is generated by **sdlgendata** to the **dsilog** program. The syntax is:
sdlgendata logfile_set class | dsilog logfile_set class -vo
2. Check the output to see if your class specification file matches the format of your data collection process. If the **sdlgendata** program outputs something different from your program, you have either an error in your output format or an error in the class specification file.
3. Before you begin collecting real data, delete all log files from the testing process.

Logging Data to the Log File Set

After you have created the log file set, and optionally tested it, update Performance Collection Component configuration files as needed, and then run the **dsilog** program to log incoming data.

1. Update the data source configuration file, **datasources**, to add the DSI log files as data sources for generating alarms.
2. Modify the alarm definitions file, **alarmdef**, if you want to alarm on specific DSI metrics. For more information, see [Defining Alarms for DSI Metrics](#).
3. Optionally, test the logging process by piping data (which may be generated by **sdlgendata** to match your class specification) to the **dsilog** program with the **-vi** option set.
4. Check the data to be sure it is being correctly logged.
5. After testing, remove the data that was tested.
6. Start the collection process from the command line.
7. Pipe the data from the collection process to **dsilog** (or some other way to get it to **stdin**) with the appropriate variables and options set. For example:

```
<program or process with variables>| dsilog logfile_set class
```

Note: The **dsilog** program is designed to receive a continuous stream of data. Therefore, it is important to structure scripts so that **dsilog** receives continuous input data. Do not write scripts that create a new **dsilog** process for new input data points. This can cause duplicate timestamps to be written to the **dsilog** file, and can cause problems for Performance Manager and **perfarm** when reading the file.

See [Examples of Data Source Integration](#), for examples of problematic and recommended scripts.

For more information about **dsilog** options, see [dsilog Logging Process](#).

Using the Logged Data

Once you have created the DSI log files, you can export the data using the Performance Collection Component's **extract** program. You can also configure alarms to occur when DSI metrics exceed defined conditions.

Here are ways to use logged DSI data:

- Export the data for use in reporting tools such as spreadsheets.
- Display exported DSI data using analysis tools such as in Performance Manager.
- Monitor alarms using HP Operations Manager or HP Network Node Manager.

Note: You cannot create extracted log files from DSI log files.

Chapter 15: DSI Class Specification Reference

This chapter provides detailed reference information about:

- Class specifications
- Class specifications syntax
- Metrics descriptions in the class specifications

Class Specifications

For each source of incoming data, you must create a class specification file to describe the format for storing incoming data. To create the file, use the class specification language described in the next section, [Class Specification Syntax](#). The class specification file contains:

- a class description, which assigns a name and numeric ID to the incoming data set, determines how much data will be stored, and specifies when to roll data to make room for new data.
- metric descriptions for each individual data item. A metric description names and describes a data item. It also specifies the summary level to apply to data (RECORDS PER HOUR) if more than one record arrives in the time interval that is configured for the class.

To generate the class specification file, use any editor or word processor that lets you save the file as an ASCII text file. You specify the name of the class specification file when you run `sd1comp` to compile it. When the class specification is compiled, it automatically creates or updates a log file set for storage of the data.

The class specification allows you to determine how many records per hour will be stored for the class, and to specify a summarization method to be used if more records arrive than you want to store. For instance, if you have requested that 12 records per hour be stored (a record every five minutes) and records arrive every minute, you could have some of the data items averaged and others totaled to maintain a running count.

Note: The DSI compiler, `sd1comp`, creates files with the following names for a log file set (named `logfile_set_name`):

`logfile_set_name` and `logfile_set_name.desc`

`sd1comp` creates a file with the following default name for a class (named `class_name`):

`logfile_set_name.class_name`

Avoid the use of class specification file names that conflict with these naming conventions, or `sd1comp` will fail.

Class Specification Syntax

Syntax statements shown in brackets [] are optional. Multiple statements shown in braces { } indicate that one of the statements must be chosen. Italicized words indicate a variable name or number you enter. Commas can be used to separate syntax statements for clarity anywhere except directly preceding the semicolon, which marks the end of the class specification and the end of each metric specification. Statements are not case-sensitive.

Note: User-defined descriptions, such as *metric_label_name* or *class_label_name*, cannot be the same as any of the keyword elements of the DSI class specification syntax.

Comments start with # or //. Everything following a # or // on a line is ignored. Note the required semicolon after the class description and after each metric description. Detailed information about each part of the class specification and examples follow.

```
CLASS class_name = class_id_number
[ LABEL "class_label_name" ]
  [ INDEX BY { HOUR | DAY | MONTH } MAX INDEXES number
  [ [ ROLL BY { HOUR | DAY | MONTH } [ ACTION "action" ]
  [ CAPACITY { maximum_record_number } ]
[ RECORDS PER HOUR number ]
;
METRICS
metric_name = metric_id_number
[ LABEL "metric_label_name" ]
[ TOTALED | AVERAGED | SUMMARIZED BY metric_name ]
[ MAXIMUM metric_maximum_number ]
[ PRECISION { 0 | 1 | 2 | 3 | 4 | 5 } ]
[ TYPE TEXT LENGTH "length" ]
;
```

CLASS Description

To create a class description, assign a name to a group of metrics from a specific data source, specify the capacity of the class, and designate how data in the class will be rolled when the capacity is exceeded.

You must begin the class description with the CLASS keyword. The final parameter in the class specification must be followed by a semicolon.

Syntax

```
CLASS class_name = class_id_number  
[ LABEL "class_label_name"  
[ INDEX BY { HOUR | DAY | MONTH } MAX INDEXES number  
[ [ ROLL BY { HOUR | DAY | MONTH } [ ACTION "action" ]  
[ CAPACITY { maximum_record_number } ]  
[ RECORDS PER HOUR number ]  
;
```

Default Settings

The default settings for the class description are:

LABEL (*class_name*)

INDEX BY DAY

MAX INDEXES 9

RECORDS PER HOUR 12

To use the defaults, enter only the CLASS keyword with a *class_name* and numeric *class_id_number*.

CLASS

The class name and class ID identify a group of metrics from a specific data source.

Syntax

```
CLASS class_name = class_id_number
```

How to Use It

The *class_name* and *class_ID_number* must meet the following requirements:

- *class_name* is alphanumeric and can be up to 20 characters long. The name must start with an alphabetic character and can contain underscores (but no special characters).
- *class_ID_number* must be numeric and can be up to six digits long.
- Neither the *class_name* or the *class_ID_number* are case-sensitive.
- The *class_name* and *class_ID_number* must each be unique among all the classes you define and cannot be the same as any applications defined in the Performance Collection Component `parm` file. (For information about the `parm` file, see Chapter 2 of the *Performance Collection Component for UNIX User's Manual*.)

Example

```
CLASS VMSTAT_STATS = 10001;
```

LABEL

The class label identifies the class as a whole. It is used instead of the class name in Performance Manager.

Syntax

```
[ LABEL "class_label_name"]
```

How To Use It

The *class_label_name* must meet the following requirements:

- It must be enclosed in double quotation marks.
- It can be up to 48 characters long.
- It cannot be the same as any of the keyword elements of the DSI class specification syntax, such as CAPACITY, ACTION and so on.
- If it contains a double quotation mark, precede it with a backslash (\). For example, you would enter "\"my\" data" if the label is "my" data.
- If no label is specified, the *class_name* is used as the default.

Example

```
CLASS VMSTAT_STATS = 10001  
LABEL "VMSTAT data";
```

INDEX BY, MAX INDEXES, AND ROLL BY

INDEX BY, MAX INDEXES, and ROLL BY settings allow you to specify how to store data and when to discard it. With these settings you designate the blocks of data to store, the maximum number of blocks to store, and the size of the block of data to discard when data reaches its maximum index value.

Syntax

```
[INDEX BY {HOURL | DAY | MONTH} MAX INDEXES number]
```

```
[[ROLL BY {HOURL | DAY | MONTH} [ACTION "action"]]
```

How To Use It

INDEX BY settings allow blocks of data to be rolled out of the class when the class capacity is reached. The INDEX BY and RECORDS PER HOUR options can be used to indirectly set the capacity of the class as described later in [Controlling Log File Size](#).

The INDEX BY setting cannot exceed the ROLL BY setting. For example, INDEX BY DAY does not work with ROLL BY HOUR, but INDEX BY HOUR does work with ROLL BY DAY.

If ROLL BY is not specified, the INDEX BY setting is used. When the capacity is reached, all the records logged in the oldest roll interval are freed for reuse.

Any specified ACTION is performed before the data is discarded (rolled). This optional ACTION can be used to export the data to another location before it is removed from the class. For information about exporting data, see [DSI Program Reference](#).

Notes on Roll Actions

The UNIX command specified in the ACTION statement cannot be run in the background. Also, do not specify a command in the ACTION statement that will cause a long delay, because new data won't be logged during the delay.

If the command is more than one line long, mark the start and end of each line with double quotation marks. Be sure to include spaces where necessary inside the quotation marks to ensure that the various command line options will remain separated when the lines are concatenated.

If the command contains a double quotation mark, precede it with a backslash (\).

The ACTION statement is limited to 199 characters or less.

Within the ACTION statement, you can use macros to define the time window of the data to be rolled out of the log file. These macros are expanded by `dsilog`. You can use `PT_START` to specify the beginning of the block of data to be rolled out in UNIX time (seconds since 1/1/70 00:00:00) and `PT_END` to specify the end of the data in UNIX time. These are particularly useful when combined with the `extract` program to export the data before it is overwritten.

If a macro is used, its expanded length is used against the 199-character limit.

Examples

The following examples may help to clarify the relationship between the INDEX BY, MAX INDEXES, and the ROLL BY clauses.

The following example indirectly sets the CAPACITY to 144 records (1*12*12).

```
CLASS VMSTAT_STATS = 10001
LABEL "VMSTAT data"
INDEX BY HOUR
MAX INDEXES 12
RECORDS PER HOUR 12;
```

The following example indirectly sets the CAPACITY to 1440 records (1*12*120).

```
CLASS VMSTAT_STATS = 10001
LABEL "VMSTAT data"
INDEX BY HOUR
MAX INDEXES 12
RECORDS PER HOUR 120;
```

The following example shows ROLL BY HOUR.

```
CLASS VMSTAT_STATS = 10001
LABEL "VMSTAT data"
INDEX BY HOUR
MAX INDEXES 12
```


ROLL BY HOUR

RECORDS PER HOUR 120;

The following example causes all the data currently identified for rolling (excluding weekends) to be exported to a file called `sys.sd1` before the data is overwritten. Note that the last lines of the last example are enclosed in double quotation marks to indicate that they form a single command.

```
CLASS VMSTAT_STATS = 10001
```

```
LABEL "VMSTAT data"
```

```
INDEX BY HOUR
```

```
MAX INDEXES 12
```

```
ROLL BY HOUR
```

```
ACTION "extract -xp -l sd1_new -C SYS_STATS "
```

```
"-B $PT_START$ -E $PT_END$ -f sys.sd1, purge -we 17 "
```

```
RECORDS PER HOUR 120;
```

Other Examples

The suggested index settings below may help you to consider how much data you want to store.

INDEX BY	MAX INDEXES	Amount of Data Stored
HOUR	72	3 days
HOUR	168	7 days
HOUR	744	31 days
DAY	365	1 year
MONTH	12	1 year

The following table provides a detailed explanation of settings using ROLL BY

INDEX BY	MAX INDEXES	ROLL BY	Meaning
DAY	9	DAY	Nine days of data will be stored in the log file. Before logging day 10, day 1 is rolled out. These are the default values for index and max indexes.
HOUR	72	HOUR	72 hours (three days) of data will be stored in the log file. Before logging hour 73, hour 1 is rolled out. Thereafter, at the start of each succeeding hour, the "oldest" hour is rolled out.
HOUR	168	DAY	168 hours (seven days) of data will be stored in the log file. Before logging hour 169 (day 8), day 1 is rolled out. Thereafter, at the start of each succeeding day, the "oldest" day is rolled out.
HOUR	744	MONTH	744 hours (31 days) of data will be stored in the log file. Before logging hour 745 (day 32), month 1 is rolled out. Thereafter, before logging

INDEX BY	MAX INDEXES	ROLL BY	Meaning
			<p>hour 745, the “oldest” month is rolled out.</p> <p>For example, <i>dsilog</i> is started on April 15 and logs data through May 16 (744 hours). Before logging hour 745 (the first hour of May 17), <i>dsilog</i> will roll out the data for the month of April (April 15 - 30).</p>
DAY	30	DAY	<p>30 days of data will be stored in the log file. Before logging day 31, day 1 is rolled out. Thereafter, at the start of each succeeding day, the “oldest” month is rolled out.</p> <p>For example, <i>dsilog</i> is started on April 1 and logs data all month, then the April 1st will be rolled out when May 1st (day 31) data is to be logged.</p>
DAY	62	MONTH	<p>62 days of data will be stored in the log file. Before logging day 63, month 1 is rolled out. Thereafter, before logging day 63 the “oldest” month is rolled out.</p> <p>For example, if <i>dsilog</i> is started on March 1 and logs data for the months of March and April, there will be 61 days of data in the log file. Once <i>dsilog</i> logs May 1st data (the 62nd day), the log file will be full. Before <i>dsilog</i> can log the data for May 2nd, it will roll out the entire month of March.</p>
MONTH	2	MONTH	<p>Two months of data will be stored in the log file. Before logging the third month, month 1 is rolled out. Thereafter, at the start of each succeeding month, the “oldest” month is rolled out.</p> <p>For example, <i>dsilog</i> is started on January 1 and logs data for the months of January and February. Before <i>dsilog</i> can log the data for March, it will roll out the month of January.</p>

Controlling Log File Size

You determine how much data is to be stored in each class and how much data to discard to make room for new data.

Class capacity is calculated from INDEX BY (hour, day, or month), RECORDS PER HOUR, and MAX INDEXES. The following examples show the results of different settings.

In this example, the class capacity is 288 (24 indexes * 12 records per hour).

INDEX BY HOUR

MAX INDEXES 24

RECORDS PER HOUR 12

In this example, the class capacity is 504 (7 days * 24 hours per day * 3 records per hour).

INDEX BY DAY
MAX INDEXES 7
RECORDS PER HOUR 3

In this example, the class capacity is 14,880 (2 months * 31 days per month * 24 hours per day * 10 records per hour).

INDEX BY MONTH
MAX INDEXES 2
RECORDS PER HOUR 10

If you do not specify values for INDEX BY, RECORDS PER HOUR, and MAX INDEXES, DSI uses the defaults for the class descriptions. See “Default Settings” under [CLASS Description](#) earlier in this chapter.

The ROLL BY option lets you determine how much data to discard each time the class record capacity is reached. The setting for ROLL BY is constrained by the INDEX BY setting in that the ROLL BY unit (hour, day, month) cannot be smaller than the INDEX BY unit.

The following example illustrates how rolling occurs given the sample

INDEX BY DAY
MAX INDEXES 6
ROLL BY DAY

Example log
Day 2 - 21 records
Day 3 - 24 records
Day 4 - 21 records
Day 5 - 24 records
Day 6 - 21 records

← Space is freed when data collection reaches 6 days. On day 7, DSI rolls the oldest day's worth of data, making room for day 7 data records.

In the above example, the class capacity is limited to six days of data by the setting:

MAX INDEXES 6.

The deletion of data is set for a day's worth by the setting:

ROLL BY DAY.

When the seventh day's worth of data arrives, the oldest day's worth of data is discarded. Note that in the beginning of the logging process, no data is discarded. After the class fills up for the first time at the end of 7 days, the roll takes place once a day.

RECORDS PER HOUR

The RECORDS PER HOUR setting determines how many records are written to the log file every hour. The default number for RECORDS PER HOUR is 12 to match Performance Collection Component's measurement interval of data sampling once every five minutes (60 minutes/12 records = logging every five minutes).

The default number or the number you enter could require the logging process to summarize data before it becomes part of the log file. The method used for summarizing each data item is specified in the metric description. For more information, see [Summarization Method](#) later in this chapter.

Syntax

[RECORDS PER HOUR *number*]

How To Use It

The logging process uses this value to summarize incoming data to produce the number of records specified. For example, if data arrives every minute and you have set RECORDS PER HOUR to 6 (every 10 minutes), 10 data points are summarized to write each record to the class. Some common RECORDS PER HOUR settings are shown below:

```
RECORDS PER HOUR 6      --> 1 record/10 minutes
RECORDS PER HOUR 12     --> 1 record/5 minutes
RECORDS PER HOUR 60     --> 1 record/minute
RECORDS PER HOUR 120    --> 1 record/30 seconds
```

Notes

RECORDS PER HOUR can be overridden by the `-s seconds` option in `dsilog`. However, overriding the original setting could cause problems when Performance Manager graphs the data.

If `dsilog` receives no metric data for an entire logging interval, a missing data indicator is logged for that metric. DSI can be forced to use the last value logged with the `-asyn` option in `dsilog`. For a description of the `-asyn` option, see [dsilog Logging Process](#).

Example

In this example, a record will be written every 10 minutes.

```
CLASS VMSTAT_STATS = 10001
LABEL "VMSTAT data"
RECORDS PER HOUR 6;
```

CAPACITY

CAPACITY is the number of records to be stored in the class.

Syntax

```
[CAPACITY {maximum_record_number}]
```

How To Use It

Class capacity is derived from the setting in RECORDS PER HOUR, INDEX BY, and MAX INDEXES. The CAPACITY setting is ignored unless a capacity larger than the derived values of these other settings is specified. If this situation occurs, the MAX INDEXES setting is increased to provide the specified capacity.

Example

```
INDEX BY DAY  
MAX INDEXES 9  
RECORDS PER HOUR 12  
CAPACITY 3000
```

In the above example, the derived class capacity is 2,592 records (9 days * 24 hours per day * 12 records per hour).

Because 3000 is greater than 2592, `sdlcomp` increases MAX INDEXES to 11, resulting in the class capacity of 3168. After compilation, you can see the resulting MAX INDEXES and CAPACITY values by running `sdlutil` with the `-decomp` option.

Managing Data with sdlutil

To manage the data from a DSI log file, use the `sdlutil` program to do any of the following tasks:

- list currently defined class and metric information to `stdout`. You can redirect output to a file.
- list complete statistics for classes to `stdout`.
- show metric descriptions for all metrics listed.
- list the files in a log file set.
- remove classes and data from a log file set.
- recreate a class specification from the information in the log file set.
- display version information.

METRICS

The metric name and id number identify the metric being collected.

Syntax

```
METRICS  
metric_name = metric_id_number
```

How To Use It

The metrics section must start with the METRICS keyword before the first metric definition. Each metric must have a metric name that meets the following requirements:

- Must not be longer than 20 characters.
- Must begin with an alphabetic character.
- Can contain only alphanumeric characters and underscores.
- Is not case-sensitive.

The metric also has a metric ID number that must not be longer than 6 characters.

The *metric_name* and *metric_id_number* must each be unique among all the metrics you define in the class. The combination *class_name:metric_name* must be unique for this system, and it cannot be the same as any *application_name:metric_name*.

Each metric description is separated from the next by a semicolon (;).

You can reuse metric names from any other class whose data is stored in the same log file set if the definitions are identical as well (see [How Log Files Are Organized](#)). To reuse a metric definition that has already been defined in another class in the same log file set, specify just the *metric_name* without the *metric_id_number* or any other specifications. If any of the options are to be set differently than the previously defined metric, the metric must be given a unique name and numeric identifier and redefined.

The order of the metric names in this section of the class specification determines the order of the fields when you export the logged data. If the order of incoming data is different than the order you list in this specification or if you do not want to log all the data in the incoming data stream, see [DSI Program Reference](#) for information about how to map the metrics to the correct location.

A timestamp metric is automatically inserted as the first metric in each class. If you want the timestamp to appear in a different position in exported data, include the short form of the internally defined metric definition (DATE_TIME;) in the position you want it to appear. To omit the timestamp and use a UNIX timestamp (seconds since 1/1/70 00:00:00) that is part of the incoming data, choose the -timestamp option when starting the dsilog process.

The simplest metric description, which uses the metric name as the label and the defaults of AVERAGED, MAXIMUM 100, and PRECISION3 decimal places, requires the following description:

```
METRICS  
metric_name = metric_id_number
```

Note: You must compile each class using `sdlcomp` and then start logging the data for that class using the `dsilog` process, regardless of whether you have reused metric names.

Example

```
VM;
```

VM is an example of reusing a metric definition that has already been defined in another class in the same log file set.

LABEL

The metric label identifies the metric in Performance Manager graphs and exported data.

Syntax

```
[LABEL "metric_label_name"]
```

How To Use It

Specify a text string, surrounded by double quotation marks, to label the metric in graphs and exported data. Up to 48 characters are allowed. If no label is specified, the metric name is used to identify the metric.

Notes

If the label contains a double quotation mark, precede it with a backslash (\). For example, you would enter "\"my\" data" if the label is "my" data.

The *metric_label_name* cannot be the same as any of the keyword elements of the DSI class specification syntax such as CAPACITY, ACTION and so on.

Example

```
METRICS  
RUN_Q_PROCS = 106  
LABEL "Procs in run q";
```

Summarization Method

The summarization method determines how to summarize data if the number of records exceeds the number set in the RECORDS PER HOUR option of the CLASS section. For example, you would want to total a count of occurrences, but you would want to average a rate. The summarization method is only valid for numeric metrics.

Syntax

```
[{TOTALLED | AVERAGED | SUMMARIZED BY metric_name}]
```

How To Use It

SUMMARIZED BY should be used when a metric is not being averaged over time, but over another metric in the class. For example, assume you have defined metrics TOTAL_ORDERS and LINES_PER_ORDER. If these metrics are given to the logging process every five minutes but records are being written only once each hour, to correctly summarize LINES_PER_ORDER to be (total lines / total orders), the logging process must perform the following calculation every five minutes:

- Multiply LINES_PER_ORDER * TOTAL_ORDERS at the end of each five-minute interval and maintain the result in an internal running count of total lines.
- Maintain the running count of TOTAL_ORDERS.
- At the end of the hour, divide total lines by TOTAL_ORDERS.

To specify this kind of calculation, you would specify LINES_PER_ORDER as SUMMARIZED BY TOTAL_ORDERS.

If no summarization method is specified, the metric defaults to AVERAGED.

Example

```
METRICS  
ITEM_1_3 = 11203  
LABEL "TOTAL_ORDERS"  
TOTALLED;
```

```
ITEM_1_5 = 11205  
LABEL "LINES_PER_ORDER"  
SUMMARIZED BY ITEM_1_3;
```

PRECISION

PRECISION identifies the number of decimal places to be used for metric values. If PRECISION is not specified, it is calculated based on the MAXIMUM specified. If neither is specified, the default PRECISION value is 3. This setting is valid only for numeric metrics.

Syntax

```
[PRECISION{0|1|2|3|4|5}]
```

How To Use It

The PRECISION setting determines the largest value that can be logged. Use PRECISION 0 for whole numbers.

PRECISION	# of Decimal Places	Largest Acceptable Numbers	MAXIMUM
0	0	2,147,483,647	> 10,000
1	1	214,748,364.7	1001 to 10,000
2	2	21,474,836.47	101 to 1,000
3	3	2,147,483.647	11 to 1,000
4	4	214,748.3647	2 to 10
5	5	21,474.83647	1

Example

```
METRICS  
RUN_Q_PROCS = 106  
LABEL "Procs in run q"  
PRECISION 1;
```

TYPE TEXT LENGTH

The three keywords TYPE TEXT LENGTH specify that the metric is textual rather than numeric. Text is defined as any character other than ^d, \n, or the separator, if any.

Because the default delimiter between data items for dsilog input is blank space, you will need to change the delimiter if the text contains embedded spaces. Use the dsilog -c char option to specify a different separator as described in [DSI Program Reference](#).

Syntax

```
[TYPE TEXT LENGTH length]
```


How To Use It

The *length* must be greater than zero and less than 4096.

Notes

Summarization method, MAXIMUM, and PRECISION cannot be specified with text metrics. Text cannot be summarized, which means that `dsilog` will take the first logged value in an interval and ignore the rest.

Example

```
METRICS
text_1 = 16
LABEL "first text metric"
TYPE TEXT LENGTH 20
;
```

Sample Class Specification

```
CLASS VMSTAT_STATS = 10001
```

```
LABEL "VMSTAT data"
```

```
INDEX BY HOUR
```

```
MAX INDEXES 12
```

```
ROLL BY HOUR
```

```
RECORDS PER HOUR 120;
```

```
METRICS
```

```
RUN_Q_PROCS = 106
```

```
LABEL "Procs in run q"
```

```
PRECISION 0;
```

```
BLOCKED_PROCS = 107
```

```
LABEL "Blocked Processes"
```

```
PRECISION 0;
```

```
SWAPPED_PROCS = 108
```

LABEL "Swapped Processes"
PRECISION 0;

AVG_VIRT_PAGES = 201
LABEL "Avg Virt Mem Pages"
PRECISION 0;

FREE_LIST_SIZE = 202
LABEL "Mem Free List Size"
PRECISION 0;

PAGE_RECLAIMS = 303
LABEL "Page Reclaims"
PRECISION 0;

ADDR_TRANS_FAULTS = 304
LABEL "Addr Trans Faults"
PRECISION 0;

PAGES_PAGED_IN = 305
LABEL "Pages Paged In"
PRECISION 0;

PAGES_PAGED_OUT = 306
LABEL "Pages Paged Out"
PRECISION 0;

PAGES_FREED = 307
LABEL "Pages Freed/Sec"
PRECISION 0;

MEM_SHORTFALL = 308
LABEL "Exp Mem Shortfall"

PRECISION 0;

CLOCKED_PAGES = 309

LABEL "Pages Scanned/Sec"

PRECISION 0;

DEVICE_INTERRUPTS = 401

LABEL "Device Interrupts"

PRECISION 0;

SYSTEM_CALLS = 402

LABEL "System Calls"

PRECISION 0;

CONTEXT_SWITCHES = 403

LABEL "Context Switches/Sec"

PRECISION 0;

USER_CPU = 501

LABEL "User CPU"

PRECISION 0;

SYSTEM_CPU = 502

LABEL "System CPU"

PRECISION 0;

IDLE_CPU = 503

LABEL "Idle CPU"

PRECISION 0;

Chapter 16: DSI Program Reference

This chapter provides detailed reference information about:

- the `sdlcomp` compiler
- configuration files `datasources` and `alarmdef`
- the `dsilog` logging process
- exporting DSI data using the Performance Collection Component `extract` program
- the `sdlutil` data source management utility

sdlcomp Compiler

The `sdlcomp` compiler checks the class specification file for errors. If no errors are found, it adds the class and metric descriptions to the description file in the log file set you name. It also sets up the pointers in the log file set's root file to the log file to be used for data storage. If either the log file set or the log file does not exist, it is created by the compiler.

Note: You can put the DSI files anywhere on your system by specifying a full path in the compiler command. However, once the path has been specified, DSI log files *cannot* be moved to different directories. (SDL62 is the associated class specification error message, described in [SDL Error Messages](#). The format used by DSI for the class specification error messages is the prefix `SDL` (Self Describing Logfile), followed by the message number.

Compiler Syntax

```
sdlcomp [-maxclass value] specification_file  
[logfile_set[log file]] [options]
```

Variables and Options	Definitions
<code>-maxclass <i>value</i></code>	allows you to specify the maximum number of classes to be provided for when creating a new log file set. This option is ignored if it is used with the name of an existing log file set. Each additional class consumes about 500 bytes of disk space in overhead, whether the class is used or not. The default is 10 if <code>-maxclass</code> is not specified.
<code>specification_file</code>	is the name of the file that contains the class specification. If it is not in the current directory, it must be fully qualified.
<code>logfile_set</code>	is the name of the log file set this class should
<code>log file</code>	is the log file in the set that will contain the data for this class. If no log file is

Variables and Options	Definitions
	named, a new log file is created for the class and is named automatically.
-verbose	prints a detailed description of the compiler output to stdout.
-vers	displays version information.
-?	displays the syntax description.
-u	allows you to log more than one record per second. Use this option to log unsummarized data only.

Sample Compiler Output

Given the following command line:

```
->sdlcomp vmstat.spec sdl_new
```

the following code is sample output for a successful compile. Note that `vmstat.spec` is the sample specification file presented in the previous chapter.

```
sdlcomp
```

```
Check class specification syntax.
```

```
CLASS VMSTAT_STATS = 10001
```

```
LABEL "VMSTAT data"
```

```
INDEX BY HOUR
```

```
MAX INDEXES 12
```

```
ROLL BY HOUR
```

```
RECORDS PER HOUR 120;
```

```
METRICS
```

```
RUN_Q_PROCS = 106
```

```
LABEL "Procs in run q"
```

```
PRECISION 0;
```

```
BLOCKED_PROCS = 107
```

```
LABEL "Blocked Processes"
```

PRECISION 0;

SWAPPED_PROCS = 108

LABEL "Swapped Processes"

PRECISION 0;

AVG_VIRT_PAGES = 201

LABEL "Avg Virt Mem Pages"

PRECISION 0;

FREE_LIST_SIZE = 202

LABEL "Mem Free List Size"

PRECISION 0;

PAGE_RECLAIMS = 303

LABEL "Page Reclaims"

PRECISION 0;

ADDR_TRANS_FAULTS = 304

LABEL "Addr Trans Faults"

PRECISION 0;

PAGES_PAGED_IN = 305

LABEL "Pages Paged In"

PRECISION 0;

PAGES_PAGED_OUT = 306

LABEL "Pages Paged Out"

PRECISION 0;

PAGES_FREED = 307

LABEL "Pages Freed/Sec"

PRECISION 0;

MEM_SHORTFALL = 308

LABEL "Exp Mem Shortfall"

PRECISION 0;

CLOCKED_PAGES = 309

LABEL "Pages Scanned/Sec"

PRECISION 0;

DEVICE_INTERRUPTS = 401

LABEL "Device Interrupts"

PRECISION 0;

SYSTEM_CALLS = 402

LABEL "System Calls"

PRECISION 0;

CONTEXT_SWITCHES = 403

LABEL "Context Switches/Sec"

PRECISION 0;

USER_CPU = 501

LABEL "User CPU"

PRECISION 0;

SYSTEM_CPU = 502

LABEL "System CPU"

PRECISION 0;

IDLE_CPU = 503

LABEL "Idle CPU"

```
PRECISION 0;
```

Note: Time stamp inserted as first metric by default.

Syntax check successful.

Update SDL `sdl_new`.

Open SDL `sdl_new`

Add class `VMSTAT_STATS`.

Check class `VMSTAT_STATS`.

Class `VMSTAT_STATS` successfully added to log file set.

For explanations of error messages and recovery, see [Error Message](#).

Configuration Files

Before you start logging data, you may need to update two Performance Collection Component configuration files:

- `/var/opt/OV/conf/perf/datasources`
- `/var/opt/perf/alarmdef` — see the section below, for information about using the `alarmdef` configuration file.

Defining Alarms for DSI Metrics

You can use Performance Collection Component to define alarms on DSI metrics. These alarms notify you when DSI metrics meet or exceed conditions that you have defined. To define alarms, you specify conditions that, when met or exceeded, trigger an alert notification or action. You define alarms for data logged through DSI the same way as for other Performance Collection Component metrics — in the `alarmdef` file on the Performance Collection Component system. The `alarmdef` file is located in the `var/opt/perf/` configuration directory of Performance Collection Component.

Whenever you specify a DSI metric name in an alarm definition, it should be fully qualified; that is, preceded by the `datasource_name`, and the `class_name` as shown below:

```
datasource_name:class_name:metric_name
```

- `datasource_name` is the name you have used to configure the data source in the `datasources` file.
- `class_name` is the name you have used to identify the class in the class specification for the data source. You do not need to enter the `class_name` if the metric name is unique (not reused) in the class specification.
- `metric_name` is the data item from the class specification for the data source.

However, if you choose not to fully qualify a metric name, you need to include the USE statement in the `alarmdef` file to identify which data source to use. For more information about the USE statement, see Chapter 7, “Performance Alarms,” in the *HP Operations Agent for UNIX User's Manual*.

To activate the changes you made to the `alarmdef` file so that it can be read by the alarm generator, enter the `ovpa restart alarm` command in the command line.

For detailed information on the alarm definition syntax, how alarms are processed, and customizing alarm definitions, see Chapter 7 in the *HP Operations Agent for UNIX User's Manual*.

Alarm Processing

As data is logged by `dsilog` it is compared to the alarm definitions in the `alarmdef` file to determine if a condition is met or exceeded. When this occurs, an alert notification or action is triggered.

You can configure where you want alarm notifications sent and whether you want local actions performed. Alarm notifications can be sent to the central Performance Manager analysis system where you can draw graphs of metrics that characterize your system performance. SNMP traps can be sent to HP Network Node Manager. Local actions can be performed on the Performance Collection Component system. Alarm information can also be sent to Operations Manager.

dsilog Logging Process

The `dsilog` process requires that either devise your own program or use one that is already in existence for you to gain access to the data. You can then pipe this data into `dsilog`, which logs the data into the log file set. A separate logging process must be used for each class you define.

`dsilog` expects to receive data from `stdin`. To start the logging process, you could pipe the output of the process you are using to collect data to `dsilog` as shown in the following example.

```
vmstat 60 | dsilog logfile_set class
```

You can only have one pipe (|) in the command line. This is because with two pipes, UNIX buffering will hold up the output from the first command until 8000 characters have been written before continuing to the second command and piping out to the log file.

You could also use a `fifo` (named pipe). For example,

```
mkfifo -m 777 myfifo  
dsilog logfile_set class -i myfifo &  
vmstat 60 > myfifo &
```

The `&` causes the process to run in the background.

Note that you may need to increase the values of the UNIX kernel parameters `shmmni` and `nfllocks` if you are planning to run a large number of `dsilog` processes. `Shmmni` specifies the maximum number of shared memory segments; `nfllocks` specifies the maximum number of file locks on a system. The default value for each is 200. Each active DSI log file set uses a shared memory segment (`shmmni`) and one or more file locks (`nfllocks`). On HP-UX, you can change the settings for `shmmni` and `nfllocks` using the System Administration and Maintenance utility (SAM).

Syntax

`dsilog logfile_set class [options]`

The `dsilog` parameters and options are described on the following pages.

Table 1: dsilog parameters and options

Variables and Options	Definitions
<code>logfile_set</code>	is the name of the log file set where the data is to be stored. If it is not in the current directory, the name must be fully qualified.
<code>class</code>	is the name of the class to be logged.
<code>-asyn</code>	specifies that the data will arrive asynchronously with the <code>RECORDS PER HOUR</code> rate. If no data arrives during a logging interval, the data for the last logging interval is repeated. However, if <code>dsilog</code> has logged no data yet, the metric value logged is treated as missing data. This causes a flat line to be drawn in a graphical display of the data and causes data to be repeated in each record if the data is exported.
<code>-c char</code>	uses the specified character as a string delimiter/separator. You may not use the following as separators: decimal, minus sign, <code>^z</code> , <code>\n</code> . If there are embedded spaces in any text metrics then you must specify a unique separator using this option.
<code>-f format file</code>	names a file that describes the data that will be input to the logging process. If this option is not specified, <code>dsilog</code> derives the format of the input from the class specification with the following assumptions. See Creating a Format File later in this chapter for more information. Each data item in an input record corresponds to a metric that has been defined in the class specification. The metrics are defined in the class specification in the order in which they appear as data items in the input record. If there are more data items in an input record than there are metric definitions, <code>dsilog</code> ignores all additional data items.
<code>-fformat file (continued)</code>	If the class specification lists more metric definitions than there are input data items, the field will show "missing" data when the data is exported, and no data will be available for that metric when graphing data in the analysis software. The number of fields in the format file is limited to 100.
<code>-ififo or ASCII file</code>	indicates that the input should come from the <code>fifo</code> or ASCII file named. If this option is not used, input comes from <code>stdin</code> . If you use this method, start <code>dsilog</code> before starting your collection process. See man page <code>mkfifo</code> for more information about using a <code>fifo</code> . Also see Examples of Data Source Integration for examples.

Variables and Options	Definitions
-s seconds	<p>is the number of seconds by which to summarize the data. The -s option overrides the summarization interval and the summarization rate defaults to RECORDS PER HOUR in the class specification. If present, this option overrides the value of RECORDS PER HOUR.</p> <p>A zero (0) turns off summarization, which means that all incoming data is logged. Caution should be used with the -s 0 option because dsilog will timestamp the log data at the time the point arrived. This can cause problems for Performance Manager and perfalarm, which work best with timestamps at regular intervals. If the log file will be accessed by Performance Manager, use of the -s 0 option is discouraged.</p>
-t	prints everything that is logged to stdout in ASCII format.
- timestamp	indicates that the logging process should not provide the timestamp, but use the one already provided in the input data. The timestamp in the incoming data must be in UNIX timestamp format (seconds since 1/1/70 00:00:00) and represent the local time.
-vi	filters the input through dsilog and writes errors to stdout instead of the log file. It does not write the actual data logged to stdout (see the -vo option below). This can be used to check the validity of the input.
-vo	filters the input through dsilog and writes the actual data logged and errors to stdout instead of the log file. This can be used to check the validity of the data summarization.
-vers	displays version information
-?	displays the syntax description.

How dsilog Processes Data

The dsilog program scans each input data string, parsing delimited fields into individual numeric or text metrics. A key rule for predicting how the data will be processed is the validity of the input string. A valid input string requires that a delimiter be present between any specified metric types (numeric or text). A blank is the default delimiter, but a different delimiter can be specified with the dsilog -c char command line option.

You *must* include a new line character at the end of any record fed to DSI in order for DSI to interpret it properly.

Testing the Logging Process with Sdlgendata

Before you begin logging data, you can test the compiled log file set and the logging process using the sdlgendata program. sdlgendata discovers the metrics for a class (as described in the class specification) and generates data for each metric in a class.

Syntax

```
sdlgendata logfile_set class [options]
```

Sdlgendata parameters and options are explained below.

Table 2: Sdlgendata parameters and options

Variables and Options	Definitions
logfile_set	is the name of the log file set to generate data for.
class	is the data class to generate data for.
- timestamp [number]	provides a timestamp with the data. If a negative number or no number is supplied, the current time is used for the timestamp. If a positive number is used, the time starts at 0 and is incremented by <i>number</i> for each new data record.
- wait number	causes a wait of <i>number</i> seconds between records generated.
-cycle number	recycles data after <i>number</i> cycles.
-vers	displays version information.
-?	displays the syntax description.

By piping sdlgendata output to dsilog with either the -vi or -vo options, you can verify the input (-vi) and verify the output (-vo) before you begin logging with your own process or program.

Note: After you are finished testing, delete all log files created from the test. Otherwise, these files remain as part of the log file test.

Use the following command to pipe data from sdlgendata to the logging process. The -vi option specifies that data is discarded and errors are written to stdout. Press CTRL+C or other interrupt control character to stop data generation.

```
sdlgendata logfile_set class -wait 5 | dsilog  
\logfile_set class -s 10 -vi
```

The previous command generates data that looks like this:

```
dsilog  
I: 744996402 1.0000 2.0000 3.0000 4.0000 5.0000 6.0000 7.0000  
I: 744996407 2.0000 3.0000 4.0000 5.0000 6.0000 7.0000 8.0000  
I: 744996412 3.0000 4.0000 5.0000 6.0000 7.0000 8.0000 9.0000  
I: 744996417 4.0000 5.0000 6.0000 7.0000 8.0000 9.0000 10.0000
```

```
I: 744996422 5.0000 6.0000 7.0000 8.0000 9.0000 10.0000 11.0000
I: 744996427 6.0000 7.0000 8.0000 9.0000 10.0000 11.0000 12.0000
I: 744996432 7.0000 8.0000 9.0000 10.000 11.000 12.0000 13.0000
I: 744996437 8.0000 9.0000 10.0000 11.0000 12.0000 13.0000 14.0000
```

You can also use the `-vo` option of `dsilog` to examine input and summarized output for your real data without actually logging it. The following command pipes `vmstat` at 5-second intervals to `dsilog` where it is summarized to 10 seconds.

```
->vmstat 5 | dsilog logfile_set class -s 10 -vo
dsilog
I: 744997230 0.0000 0.0000 21.0000 2158.0000 1603.0000 2.0000 2.0000
I: 744997235 0.0000 0.0000 24.0000 2341.0000 1514.0000 0.0000 0.0000
interval marker
L: 744997230 0.0000 0.0000 22.5000 2249.5000 1558.5000 1.0000 1.0000

I: 744997240 0.0000 0.0000 23.0000 2330.0000 1513.0000 0.0000 0.0000
I: 744997245 0.0000 0.0000 20.0000 2326.0000 1513.0000 0.0000 0.0000
interval marker
L: 744997240 0.0000 0.0000 21.5000 2328.0000 1513.0000 0.0000 0.0000

I: 744997250 0.0000 0.0000 22.0000 2326.0000 1513.0000 0.0000 0.0000
I: 744997255 0.0000 0.0000 22.0000 2303.0000 1513.0000 0.0000 0.0000
interval marker
L: 744997250 0.0000 0.0000 22.0000 2314.5000 1513.0000 0.0000 0.0000

I: 744997260 0.0000 0.0000 22.0000 2303.0000 1512.0000 0.0000 0.0000
I: 744997265 0.0000 0.0000 28.0000 2917.0000 1089.0000 9.0000 33.0000
interval marker
L: 744997260 0.0000 0.0000 25.0000 2610.0000 1300.5000 4.5000 16.5000

I: 744997270 0.0000 0.0000 28.0000 2887.0000 1011.0000 3.0000 9.0000
I: 744997275 0.0000 0.0000 27.0000 3128.0000 763.0000 8.0000 6.0000
interval marker
L: 744997270 0.0000 0.0000 27.5000 3007.5000 887.0000 5.5000 12.5000
```

You can also use the `dsilog -vo` option to use a file of old data for testing, as long as the data contains its own UNIX timestamp (seconds since 1/1/70 00:00:00). To use a file of old data, enter a command like this:

```
dsilog -timestamp -vo <oldfile>
```

Creating a Format File

Create a format file to map the data input to the class specification if:

- the data input contains data that is not included in the class specification.
- incoming data has metrics in a different order than you have specified in the class specification.

A format file is an ASCII text file that you can create with `vi` or any text editor. Use the `-f` option in `dsilog` to specify the fully qualified name of the format file.

Because the logging process works by searching for the first valid character after a delimiter (either a space by default or user-defined with the `dsilog -c` option) to start the next metric, the format file simply tells the logging process which fields to skip and what metric names to associate with fields not skipped.

`$numeric` tells the logging process to skip one numeric metric field and go to the next. `$any` tells the logging process to skip one text metric field and go to the next. Note that the format file is limited to 100 fields.

For example, if the incoming data stream contains this information:

```
ABC 987 654 123 456
```

and you want to log only the first numeric field into a metric named `metric_1`, the format file would look like this:

```
$any metric_1
```

This tells the logging process to log only the information in the first numeric field and discard the rest of the data. To log only the information in the third numeric field, the format file would look like this:

```
$any $numeric $numeric metric_1
```

To log all four numeric data items, in reverse order, the format file would look like this:

```
$any metric_4 metric_3 metric_2 metric_1
```

If the incoming data stream contains the following information:

```
/users 15.9 3295 56.79% xdisk1 /dev/dsk/c0d0s*
```

and you want to log only the first text metric and the first two numeric fields into metric fields you name `text_1`, `num_1`, and `num_2`, respectively, the format file would look like this:

```
text_1 num_1 num_2
```

This tells the logging process to log only the information in the first three fields and discard the rest of the data.

To log all of the data, but discard the “%” following the third metric, the format file would look like this:

```
text_1 num_1 num_2 num_3 $any text_2 text_3
```

Since you are logging numeric fields and the “%” is considered to be a text field, you need to skip it to correctly log the text field that follows it.

To log the data items in a different order the format file would look like this:

```
text_3 num_2 num_1 num_3 $any text_2 text_1
```

Note that this will result in only the first six characters of *text_3* being logged if *text_1* is declared to be six characters long in the class specification. To log *all* of *text_3* as the first value, change the class specification and alter the data stream to allow extra space.

Changing a Class Specification

To change a class specification file, you must recreate the whole log file set as follows:

1. Stop the `dsilog` process.
2. Export the data from the existing log file using the UNIX timestamp option if you want to save it or integrate the old data with the new data you will be logging. See [Exporting DSI Data](#) later in this chapter for information on how to do this.
3. Run `sdlutil` to remove the log file set. See [Managing Data With sdlutil](#) later in this chapter for information on how to do this.
4. Update the class specification file.
5. Run `sdlcomp` to recompile the class specification.
6. Optionally, use the `-i` option in `dsilog` to integrate in the old data you exported in step 2. You may need to manipulate the data to line up with the new data using the `-f format_file` option.
7. Run `dsilog` to start logging based on the new class specification.
8. As long as you have not changed the log file set name or location, you do not need to update the `datasources` file.

Exporting DSI Data

To export the data from a DSI log file, use the Performance Collection Component `extract` program's `export` function. See Chapters 5 and 6 of the *HP Operations Agent for UNIX User's Manual* for details on how to use `extract` to export data. An example of exporting DSI data using command line arguments is provided on the following page.

There are several ways to find out what classes and metrics can be exported from the DSI log file. You can use `sdlutil` to list this information as described in [Managing Data with sdlutil](#) later in this chapter. Or you can use the `extract guide` command to create an export template file that lists the classes and metrics in the DSI log file. You can then use `vi` to edit, name, and save the file. The export template file is used to specify the export format, as described in Chapters 5 and 6 of the *HP Operations Agent for UNIX User's Manual*.

Note: You *must* be root or the creator of the log file to export DSI log file data.

Example of Using Extract to Export DSI Log File Data

```
extract -xp -l logfile_set -C class [options]
```

You can use `extract` command line options to do the following:

- Specify an export output file.
- Set begin and end dates and times for the first and last intervals to export.
- Export data only between certain times (shifts).
- Exclude data for certain days of the week (such as weekends).
- Specify a separation character to put between metrics on reports.
- Choose whether or not to display headings and blank records for intervals when no data arrives and what the value displayed should be for missing or null data.
- Display exported date/time in UNIX format or date and time format.
- Set additional summarization levels.

Viewing Data in Performance Manager

In order to display data from a DSI log file in Performance Manager, you need to configure the DSI log file as a Performance Collection Component data source. Before you start logging data, configure the data source by adding it to the `datasources` file on the Performance Collection Component system.

You can centrally view, monitor, analyze, compare, and forecast trends in DSI data using Performance Manager. Performance Manager helps you identify current and potential problems. It provides the information you need to resolve problems before user productivity is affected.

Managing Data with sdlutil

To manage the data from a DSI log file, use the `sdlutil` program to do any of the following tasks:

- list currently defined class and metric information to `stdout`. You can redirect output to a file.
- list complete statistics for classes to `stdout`.
- show metric descriptions for all metrics listed.
- list the files in a log file set.
- remove classes and data from a log file set.
- recreate a class specification from the information in the log file set.
- display version information.

Syntax

```
sdlutil logfile_set [option]
```


Variables and Options	Definitions
logfile_set	is the name of a log file set created by compiling a class specification.
-classes <i>classlist</i>	provides a class description of all classes listed. If none are listed, all are provided. Separate the Items in the <i>classlist</i> with spaces.
-stats <i>classlist</i>	provides complete statistics for all classes listed. If none are listed, all are provided. Separate the Items in the <i>classlist</i> with spaces.
-metrics <i>metriclist</i>	provides metric descriptions for all metrics in the <i>metriclist</i> . If none are listed, all are provided. Separate the Items in the <i>metriclist</i> with spaces.
-id	displays the shared memory segment ID used by the log file.
-files	lists all the files in the log file set.
-rm all	removes all classes and data as well as their data and shared memory ID from the log file.
-decomp <i>classlist</i>	recreates a class specification from the information in the log file set. The results are written to <code>stdout</code> and should be redirected to a file if you plan to make changes to the file and reuse it. Separate the Items in the <i>classlist</i> with spaces.
-vers	displays version information.
-?	displays the syntax description.

Chapter 17: Examples of Data Source Integration

Data source integration is a very powerful and very flexible technology. Implementation of DSI can range from simple and straightforward to very complex.

This chapter contains examples of using DSI for the following tasks:

- writing a **dsilog** script
- logging vmstat data
- logging sar data
- logging who word count

Writing a dsilog Script

The **dsilog** code is designed to receive a continuous stream of data rows as input. This stream of input is summarized by **dsilog** according to the specification directives for each class, and one summarized data row is logged per requested summarization interval. Performance Manager and perfalarm work best when the timestamps written in the log conform to the expected summarization rate (records per hour). This happens automatically when **dsilog** is allowed to do the summarization.

dsilog process for each arriving input row, which may cause problems with Performance Manager and perfalarm. This method is not recommended.

- Problematic **dsilog** script
- Recommended **dsilog** script

Example 1 - Problematic dsilog Script

In the following script, a new **dsilog** process is executed for each arriving input row.

```
while :
do
    feed_one_data_row | dsilog sdlname classname
    sleep 50
done
```

Example 2 - Recommended dsilog Script

In the following script, one **dsilog** process receives a continuous stream of input data. `feed_one_data_row` is written as a function, which provides a continuous data stream to a single **dsilog** process.

```
# Begin data feed function
feed_one_data_row()
{
  while :
  do
# Perform whatever operations necessary to produce one row
# of data for feed to a dsilog process
    sleep 50
  done
}
# End data feed function

# Script mainline code
feed_one_data_row | dsilog sdlname classname
```

Logging vmstat Data

This example shows you how to set up data source integration using default settings to log the first two values reported by vmstat. You can either read this section as an overview of how the data source integration process works, or perform each task to create an equivalent DSI log file on your system.

The procedures needed to implement data source integration are:

- Creating a class specification file.
- Compiling the class specification file.
- Starting the **dsilog** logging process.

Creating a Class Specification File

The class specification file is a text file that you create to describe the class, or set of incoming data, as well as each individual number you intend to log as a metric within the class. The file can be created with the text editor of your choice. The file for this example of data source integration should be created in the **/tmp/** directory.

The following example shows the class specification file required to describe the first two vmstat numbers for logging in a class called VMSTAT_STATS. Because only two metrics are defined in this class, the logging process ignores the remainder of each vmstat output record. Each line in the file is explained in the comment lines that follow it.

```
CLASS VMSTAT_STATS = 10001;

# Assigns a unique name and number to vmstat class data

# The semicolon is required to terminate the class section
# of the file.

METRICS

# Indicates that everything that follows is a description
# of a number (metric) to be logged.

RUN_Q_PROCS = 106;

# Assigns a unique name and number to a single metric.
# The semicolon is required to terminate each metric.

BLOCKED_PROCS = 107;

# Assigns a unique name and number to another metric.
# The semicolon is required to terminate each metric.
```

Compiling the Class Specification File

When you compile the class specification file using `sdlcomp`, the file is checked for syntax errors. If none are found, `sdlcomp` creates or updates a set of log files to hold the data for the class.

Use the file name you gave to the class specification file and then specify a name for *logfile_set_name* that makes it easy to remember what kind of data the log file contains. In the command and compiler output example below, `/tmp/vmstat.spec` is used as the file name and `/tmp/VMSTAT_DATA` is used for the log file set.

```
-> sdlcomp /tmp/vmstat.spec /tmp/VMSTAT_DATA
```

```
sdlcomp X.01.04
```

```
Check class specification syntax.
```

```
CLASS VMSTAT_STATS = 10001;
```

```
METRICS
```

```
RUN_Q_PROCS = 106;
```

```
BLOCKED_PROCS = 107;
```

NOTE: Time stamp inserted as first metric by default.

Syntax check successful.

Update SDL VMSTAT_DATA.

Shared memory ID used by vmstat_data=219

Class VMSTAT_STATS successfully added to log file set.

This example creates a log file set called VMSTAT_DATA in the `/tmp/` directory, which includes a root file and description file in addition to the data file. The log file set is ready to accept logged data. If there are syntax errors in the class specification file, messages indicating the problems are displayed and the log file set is not created.

Starting the dsilog Logging Process

Now you can pipe the output of vmstat directly to the **dsilog** logging process. Use the following command:

```
vmstat 60 | dsilog /tmp/VMSTAT_DATA VMSTAT_STATS &
```

This command runs `vmstat` every 60 seconds and sends the output directly to the VMSTAT_STATS class in the VMSTAT_DATA log file set. The command runs in the background. You could also use `remsh` to feed `vmstat` in from a remote system.

Note that the following message is generated at the start of the logging process:

```
Metric null has invalid dataIgnore to end of line, metric value exceeds maximum
```

This message is a result of the header line in the `vmstat` output that **dsilog** cannot log. Although the message appears on the screen, **dsilog** continues to run and begins logging data with the first valid input line.

Accessing the Data

You can use the `sdlutil` program to report on the contents of the class:

```
sdlutil /tmp/VMSTAT_DATA -stats VMSTAT_STATS
```

Note: By default, data will be summarized and logged once every five minutes.

You can use **extract** program command line arguments to export data from the class. For example:

```
extract -xp -l /tmp/VMSTAT_DATA -C VMSTAT_STATS -ut -f stdout
```

Note that to export DSI data, you must be root or the creator of the log file.

Logging sar Data from One File

This example shows you how to set up several DSI data collections using the standard sar (system activity report) utility to provide the data.

When you use a system utility, it is important to understand exactly how that utility reports the data. For example, note the difference between the following two sar commands:

```
sar -u 1 1  
  
HP-UX hpptc99 A.11.00 E 9000/855 04/10/99
```

```
10:53:15 %usr %sys %wio %idle  
10:53:16 2 7 6 85
```

```
sar -u 5 2  
  
HP-UX hpptc99 A.11.00 E 9000/855 04/10/99
```

```
10:53:31 %usr %sys %wio %idle  
10:53:36 4 5 0 91 10:53:41 0 0 0 99
```

```
Average 2 2 0 95
```

As you can see, specifying an iteration value greater than 1 causes sar to display an average across the interval. This average may or may not be of interest but can affect your DSI class specification file and data conversion. You should be aware that the output of sar, or other system utilities, may be different when executed on different UNIX platforms. You should become very familiar with the utility you are planning to use before creating your DSI class specification file.

Our first example uses sar to monitor CPU utilization via the -u option of sar. If you look at the man page for sar, you will see that the -u option reports the portion of time running in user mode (%usr), running in system mode (%sys), idle with some process waiting for block I/O (%wio), and otherwise idle (%idle). Because we are more interested in monitoring CPU activity over a long period of time, we use the form of sar that does not show the average.

Creating a Class Specification File

The first task to complete is the creation of a DSI class specification file. The following is an example of a class specification that can be used to describe the incoming data:

```
# sar_u.spec  
#  
# sar -u class definition for HP systems.  
#  
# ==> 1 minute data; max 24 hours; indexed by hour; roll by day  
#
```

```
CLASS sar_u = 1000  
LABEL "sar -u data"  
INDEX BY hour  
MAX INDEXES 24  
ROLL BY day  
ACTION "./sar_u_roll $PT_START$ $PT_END$"  
RECORDS PER HOUR 60  
;
```

METRICS

```
hours_1 = 1001  
LABEL "Collection Hour"  
PRECISION 0  
;
```

```
minutes_1 = 1002  
LABEL "Collection Minute"  
PRECISION 0  
;
```

```
seconds_1 = 1003  
LABEL "Collection Second"  
PRECISION 0  
;
```

```
user_cpu = 1004  
LABEL "%user"  
AVERAGED  
MAXIMUM 100  
PRECISION 0  
;
```

```
sys_cpu = 1005  
LABEL "%sys"  
AVERAGED  
MAXIMUM 100  
PRECISION 0  
;
```

```
wait_IO_cpu = 1006  
LABEL "%wio"  
AVERAGED  
MAXIMUM 100  
PRECISION 0  
;
```

```
idle_cpu = 1007  
LABEL "%idle"  
AVERAGED  
MAXIMUM 100  
PRECISION 0  
;
```

Compiling the Class Specification File

The next task is to compile the class specification file using the following command.

```
sdlcomp sar_u.spec sar_u_log
```

The output of the `sar -u` command is a system header line, a blank line, an option header line, and a data line consisting of a time stamp followed by the data we want to capture. The last line is the only line that is interesting. So, from the `sar -u` command, we need a mechanism to save only the last line of output and feed that data to DSI.

dsilog expects to receive data from **stdin**. To start the logging process, you could pipe output from the process you are using to **dsilog**. However, you can only have one pipe (|) in the command line. When two pipes are used, UNIX buffering retains the output from the first command until 8000 characters have been written before continuing to the second command and piping out to the log file. As a result, doing something like the following does not work:

```
sar -u 60 1 | tail -1 | dsilog
```

Therefore, we use a *fifo* as the input source for DSI. However, this is not without its problems.

Assume we were to use the following script:

```
#!/bin/ksh sar_u_feed
# sar_u_feed script that provides sar -u data to DSI via
# a fifo(sar_u.fifo)
while : # (infinite loop)
do
# specify a one minute interval using tail to extract the
# last sar output record(contains the time stamp and data),
# saving the data to a file./usr/bin/sar -u 60 1 2>/tmp/dsierr | tail -1 >
/usr/tmp/sar_u_data
# Copy the sar data to the fifo that the dsilog process is
# reading.

cat /usr/tmp/sar_u_data > ./sar_u.fifo
done
```

Unfortunately, this script will not produce the desired results if run as is. This is because the `cat` command opens the `fifo`, writes the data record, and then closes the `fifo`. The close indicates to **dsilog** that there is no more data to be written to the log, so **dsilog** writes this one data record and terminates. What is needed is a dummy process to “hold” the `fifo` open. Therefore, we need a dummy `fifo` and a process that opens the dummy `fifo` for input and the `sar_u.fifo` for output. This will hold the `sar_u.fifo` open, thereby preventing **dsilog** from terminating.

Starting the DSI Logging Process

Now let's take a step by step approach to getting the `sar -u` data to **dsilog**.

1. Create two `fifos`; one is the dummy `fifo` used to “hold open” the real input `fifo`.
Dummy `fifo`.
`mkfifo ./hold_open.fifo`
Real input `fifo` for `dsilog`

```
mkfifo ./sar_u.fifo
```

2. Start **dsilog** using the `-i` option to specify the input coming from a fifo. It is important to start **dsilog** before starting the sar data feed (`sar_u_feed`).

```
dsilog ./sar_u_log sar_u \  
-i ./sar_u.fifo &
```

3. Start the dummy process to hold open the input fifo.

```
cat ./hold_open.fifo \  
> ./sar_u.fifo &
```

4. Start the sar data feed script (`sar_u_feed`).

```
./sar_u_feed &
```

5. The `sar_u_feed` script will feed data to **dsilog** until it is killed or the cat that holds the fifo open is killed. Our class specification file states that `sar_u_log` will be indexed by hour, contain a maximum of 24 hours, and at the start of the next day (roll by day), the script `sar_u_roll` will be executed.

```
#!/bin/ksh sar_u_rol  
!  
# Save parameters and current date in sar_u_log_roll_file.  
# (Example of adding comments/other data to the roll file).
```

```
mydate=`date`  
echo "$# $0 $1 $2" >> ./sar_u_log_roll_file  
echo $mydate >> ./sar_u_log_roll_file
```

```
extract -l ./sar_u_log -C sar_u -B $1 -E $2 -1 -f \  
stdout -xp >> ./sar_u_log_roll_file
```

6. The roll script saves the data being rolled out in an ASCII text file that can be examined with a text editor or printed to a printer.

Logging sar Data from Several Files

If you are interested in more than just CPU utilization, you can either have one class specification file that describes the data, or have a class specification file for each option and compile these into one log file set. The first example shows separate class specification files compiled into a single log file set.

In this example, we will monitor CPU utilization, buffer activity (`sar -b`), and system calls (`sar -c`). Logging data in this manner requires three class specification files, three **dsilog** processes, three **dsilog** input fifos, and three scripts to provide the sar data.

Creating Class Specification Files

The following are the class specification files for each of these options.

```
# sar_u_mc.spec
#
# sar -u class definition for log files on HP systems.
#
# ==> 1 minute data; max 24 hours; indexed by hour; roll by day
#
CLASS sar_u = 1000
LABEL "sar -u data"
INDEX BY hour
MAX INDEXES 24
ROLL BY day
ACTION "./sar_u_mc_roll $PT_START$ $PT_END$"
RECORDS PER HOUR 60
;

METRICS
hours_1 = 1001
LABEL "Collection Hour"
PRECISION 0
;

minutes_1 = 1002
LABEL "Collection Minute"
PRECISION 0
;

seconds_1 = 1003
LABEL "Collection Second"
PRECISION 0
;
```

```
user_cpu = 1004  
LABEL "%user"  
AVERAGED  
MAXIMUM 100  
PRECISION 0  
;
```

```
sys_cpu = 1005  
LABEL "%sys"  
AVERAGED  
MAXIMUM 100  
PRECISION 0  
;
```

```
wait_IO_cpu = 1006  
LABEL "%wio"  
AVERAGED  
MAXIMUM 100  
PRECISION 0  
;
```

```
idle_cpu = 1007  
LABEL "%idle"  
AVERAGED  
MAXIMUM 100  
PRECISION 0  
;
```

```
# sar_b_mc.spec  
#  
# sar -b class definition for log files on HP systems.  
#  
# ==> 1 minute data; max 24 hours; indexed by hour; roll by day  
#
```

```
CLASS sar_b = 2000  
LABEL "sar -b data"  
INDEX BY hour  
MAX INDEXES 24  
ROLL BY day  
ACTION "./sar_b_mc_roll $PT_START$ $PT_END$"  
RECORDS PER HOUR 60  
;
```

```
METRICS  
hours_2 = 2001  
LABEL "Collection Hour"  
PRECISION 0  
;
```

```
minutes_2 = 2002  
LABEL "Collection Minute"  
PRECISION 0  
;
```

```
seconds_2 = 2003  
LABEL "Collection Second"  
PRECISION 0  
;
```

```
bread_per_sec = 2004  
LABEL "bread/s"  
PRECISION 0  
;
```

```
lread_per_sec = 2005  
LABEL "lread/s"  
PRECISION 0
```

;

read_cache = 2006

LABEL "%rcache"

MAXIMUM 100

PRECISION 0

;

bwrit_per_sec = 2007

LABEL "bwrit/s"

PRECISION 0

;

lwrit_per_sec = 2008

LABEL "lwrit/s"

PRECISION 0

;

write_cache = 2009

LABEL "%wcache"

MAXIMUM 100

PRECISION 0

;

pread_per_sec = 2010

LABEL "pread/s"

PRECISION 0

;

pwrit_per_sec = 2011

LABEL "pwrit/s"

PRECISION 0

;

```
# sar_c_mc.spec
#
# sar -c class definition for log files on HP systems.
#
# ==> 1 minute data; max 24 hours; indexed by hour; roll by day
#

CLASS sar_c = 5000
LABEL "sar -c data"
INDEX BY hour
MAX INDEXES 24
ROLL BY day
ACTION "./sar_c_mc_roll $PT_START$ $PT_END$"
RECORDS PER HOUR 60
;

METRICS
hours_5 = 5001
LABEL "Collection Hour"
PRECISION 0
;

minutes_5 = 5002
LABEL "Collection Minute"
PRECISION 0
;

seconds_5 = 5003
LABEL "Collection Second"
PRECISION 0
;

scall_per_sec = 5004
LABEL "scall/s"
```

```
PRECISION 0
```

```
;
```

```
sread_per_sec = 5005
```

```
LABEL "sread/s"
```

```
PRECISION 0
```

```
;
```

```
swrit_per_sec = 5006
```

```
LABEL "swrit/s"
```

```
PRECISION 0
```

```
;
```

```
fork_per_sec = 5007
```

```
LABEL "fork/s"
```

```
PRECISION 2
```

```
;
```

```
exec_per_sec = 5008
```

```
LABEL "exec/s"
```

```
PRECISION 2
```

```
;
```

```
rchar_per_sec = 5009
```

```
LABEL "rchar"
```

```
PRECISION 0
```

```
;
```

```
wchar_per_sec = 5010
```

```
LABEL "wchar/s"
```

```
PRECISION 0
```

```
;
```


The following are the two additional scripts that are needed to supply the sar data.

```
#!/bin/ksh

# sar_b_feed script that provides sar -b data to DSI via
# a fifo (sar_b.fifo)
while : # (infinite loop)
do

# specify a one minute interval using tail to extract the
# last sar output record(contains the time stamp and data),
# saving the data to a file.

/usr/bin/sar -b 60 1 2>/tmp/dsierr | tail -1 &> \
/usr/tmp/sar_b_data

# Copy the sar data to the fifo that the dsilog process is reading.
cat /usr/tmp/sar_b_data > ./sar_b.fifo
done

#!/bin/ksh sar_c_feed

# sar_c_feed script that provides sar -c data to DSI via
# a fifo(sar_c.fifo)
while : # (infinite loop)
do

# specify a one minute interval using tail to extract the
# last sar output record(contains the time stamp and data),
# saving the data to a file.

/usr/bin/sar -c 60 1 2>/tmp/dsierr | tail -1 > /usr/tmp/sar_c_data

# Copy the sar data to the fifo that the dsilog process is reading.
cat /usr/tmp/sar_c_data > ./sar_c.fifo
```

done

Compiling the Class Specification Files

Compile the three specification files into one log file set:

```
sdlcomp ./sar_u_mc.spec sar_mc_log
sdlcomp ./sar_b_mc.spec sar_mc_log
sdlcomp ./sar_c_mc.spec sar_mc_log
```

Starting the DSI Logging Process

Returning to the step by step approach for the sar data:

1. Create four fifos; one will be the dummy fifo used to “hold open” the three real input fifos.
Dummy fifo.mkfifo ./hold_open.fifo
sar -u input fifo for dsilog.mkfifo ./sar_u.fifo
sar -b input fifo for dsilog.mkfifo ./sar_b.fifo
sar -c input fifo for dsilog.mkfifo ./sar_c.fifo
2. Start **dsilog** using the **-i** option to specify the input coming from a fifo. It is important to start **dsilog** before starting the sar data feeds.
dsilog ./sar_mc_log sar_u \
-i ./sar_u.fifo &

dsilog ./sar_mc_log sar_b \
-i ./sar_b.fifo &

dsilog ./sar_mc_log sar_c \
-i ./sar_c.fifo &
3. Start the dummy process to hold open the input fifo.
cat ./hold_open.fifo \
> ./sar_u.fifo &
cat ./hold_open.fifo \
> ./sar_b.fifo &
cat ./hold_open.fifo \
> ./sar_c.fifo &
4. Start the sar data feed scripts.
./sar_u_feed &
./sar_b_feed &
./sar_c_feed &

Logging sar Data for Several Options

The last example for using sar to supply data to DSI uses one specification file to define the data from several sar options (ubycwvm).

```
# sar_ubycwvm.spec
#
# sar -ubycwvm class definition for HP systems.
#
# ==> 1 minute data; max 24 hours; indexed by hour; roll by day#

CLASS sar_ubycwvm = 1000
LABEL "sar -ubycwvm data"
INDEX BY hour
MAX INDEXES 24ROLL BY day
ACTION "./sar_ubycwvm_roll $PT_START$ $PT_END$"
RECORDS PER HOUR 60
;

METRICShours = 1001
LABEL "Collection Hour"
PRECISION 0
;

minutes = 1002
LABEL "Collection Minute"
PRECISION 0
;

seconds = 1003
LABEL "Collection Second"
PRECISION 0
;

user_cpu = 1004
```

```
LABEL "%user"
```

```
AVERAGED
```

```
MAXIMUM 100
```

```
PRECISION 0
```

```
;
```

```
sys_cpu = 1005
```

```
LABEL "%sys"
```

```
AVERAGED
```

```
MAXIMUM 100
```

```
PRECISION 0
```

```
;
```

```
wait_IO_cpu = 1006
```

```
LABEL "%wio"
```

```
AVERAGED
```

```
MAXIMUM 100
```

```
PRECISION 0
```

```
;
```

```
idle_cpu = 1007
```

```
LABEL "%idle"
```

```
AVERAGED
```

```
MAXIMUM 100
```

```
PRECISION 0
```

```
;
```

```
bread_per_sec = 1008
```

```
LABEL "bread/s"
```

```
PRECISION 0
```

```
;
```

```
lread_per_sec = 1009
```

```
LABEL "lread/s"
```

```
PRECISION 0
```

```
;
```

```
read_cache = 1010
```

```
LABEL "%rcache"
```

```
MAXIMUM 100
```

```
PRECISION 0
```

```
;
```

```
bwrit_per_sec = 1011
```

```
LABEL "bwrit/s"
```

```
PRECISION 0
```

```
;
```

```
lwrit_per_sec = 1012
```

```
LABEL "lwrit/s"
```

```
PRECISION 0
```

```
;
```

```
write_cache = 1013
```

```
LABEL "%wcache"
```

```
MAXIMUM 100
```

```
PRECISION 0
```

```
;
```

```
pread_per_sec = 1014
```

```
LABEL "pread/s"
```

```
PRECISION 0
```

```
;
```

```
pwrit_per_sec = 1015
```

```
LABEL "pwrit/s"
```

```
PRECISION 0
```

```
;
```

```
rawch = 1016  
LABEL "rawch/s"  
PRECISION 0  
;
```

```
canch = 1017  
LABEL "canch/s"  
PRECISION 0  
;
```

```
outch = 1018  
LABEL "outch/s"  
PRECISION 0  
;
```

```
rcvin = 1019  
LABEL "rcvin/s"  
PRECISION 0  
;
```

```
xmtin = 1020  
LABEL "xmtin/s"  
PRECISION 0  
;
```

```
mdmin = 1021  
LABEL "mdmin/s"  
PRECISION 0  
;
```

```
scall_per_sec = 1022  
LABEL "scall/s"  
PRECISION 0
```

;

sread_per_sec = 1023

LABEL "sread/s"

PRECISION 0

;

swrit_per_sec = 1024

LABEL "swrit/s"

PRECISION 0

;

fork_per_sec = 1025

LABEL "fork/s"

PRECISION 2

;

exec_per_sec = 1026

LABEL "exec/s"

PRECISION 2

;

rchar_per_sec = 1027

LABEL "rchar/s"

PRECISION 0

;

wchar_per_sec = 1028

LABEL "wchar/s"

PRECISION 0

;

swpin = 1029

LABEL "swpin/s"

```
PRECISION 2
```

```
;
```

```
bswin = 1030
```

```
LABEL "bswin/s"
```

```
PRECISION 1
```

```
;
```

```
swpot = 1031
```

```
LABEL "swpot/s"
```

```
PRECISION 2
```

```
;
```

```
bswot = 1032
```

```
LABEL "bswot/s"
```

```
PRECISION 1
```

```
;
```

```
blks = 1033
```

```
LABEL "pswch/s"
```

```
PRECISION 0
```

```
;
```

```
iget_per_sec = 1034
```

```
LABEL "iget/s"
```

```
PRECISION 0
```

```
;
```

```
namei_per_sec = 103
```

```
5LABEL "namei/s"
```

```
PRECISION 0
```

```
;
```

```
dirbk_per_sec = 1036
```



```
LABEL "dirbk/s"
```

```
PRECISION 0
```

```
;
```

```
num_proc = 1037
```

```
LABEL "num proc"
```

```
PRECISION 0
```

```
;
```

```
proc_tbl_size = 1038
```

```
LABEL "proc tbl size"
```

```
PRECISION 0
```

```
;
```

```
proc_ov = 1039
```

```
LABEL "proc ov"
```

```
PRECISION 0
```

```
;
```

```
num_inode = 1040
```

```
LABEL "num inode"
```

```
PRECISION 0
```

```
;
```

```
inode_tbl_sz = 1041
```

```
LABEL "inode tbl sz"
```

```
PRECISION 0
```

```
;
```

```
inode_ov = 1042
```

```
LABEL "inode ov"
```

```
PRECISION 0
```

```
;
```

```
num_file = 1043  
LABEL "num file"  
PRECISION 0  
;  
  
file_tbl_sz = 1044  
LABEL "file tbl sz"  
PRECISION 0  
;  
  
file_ov = 1045  
LABEL "file ov"  
PRECISION 0  
;  
  
msg_per_sec = 1046  
LABEL "msg/s"  
PRECISION 2  
;  
  
LABEL "sema/s"  
PRECISION 2  
;
```

At this point, we need to look at the output generated from
sar -ubycwvm 1 1:

HP-UX hpptc16 A.09.00 E 9000/855 04/11/95

```
12:01:41 %usr %sys %wio %idle  
bread/s lread/s %rcache bwrit/s lwrit/s %wcache pread/s  
pwrit/s  
rawch/s canch/s outh/s cvin/s xmtin/s madmin/s  
scall/s sread/s swrit/s fork/s exec/s rchar/s wchar/s  
swpin/s bswin/s swpot/s bswot/s pswch/s  
iget/s namei/s dirbk/s
```

```
text-sz ov proc-sz ov inod-sz ov file-sz ov  
msg/s sema/s
```

```
12:01:42 22 48 30 0 0  
342 100 33 81 59 0 0  
0 0 470 0 0 0  
801 127 71 1.00 1.00 975872 272384  
0.00 0.0 0.00 0.0 251  
28 215 107  
N/A N/A 131/532 0 639/644 0 358/1141 0  
40.00 0.00
```

This output looks similar to the `sar -u` output with several additional lines of headers and data. We will again use `tail` to extract the lines of data, but we need to present this as “one” data record to **dsilog**. The following script captures the data and uses the `tr` (translate character) utility to “strip” the line feeds so **dsilog** will see it as one single line of input data.

```
#!/bin/ksh Sar_ubycwvm_feed#  
Script that provides sar data to DSI via a fifo(sar_data.fifo)  
while : # (infinite loop)  
do  
  
# specify a one minute interval using tail to extract the  
# last sar output records (contains the time stamp and data)  
# and pipe that data to tr to strip the new lines converting  
# the eight lines of output to one line of output.  
/usr/bin/sar -ubycwvm 60 1 2>/tmp/dsierr | tail -8 | \  
tr "\012" " " > /usr/tmp/sar_data  
  
# Copy the sar data to the fifo that the dsilog process is reading.  
cat /usr/tmp/sar_data > ./sar_data.fifo  
  
# Print a newline on the fifo so that DSI knows that this is  
# the end of the input record.  
  
print "\012" > ./sar_data.fifo  
done
```

The step-by-step process follows that for the earlier `sar -u` example with the exception of log file set names, class names, fifo name (`sar_ubycwvm.fifo`), and the script listed above to provide the `sar` data.

Logging the Number of System Users

The next example uses `who` to monitor the number of system users. Again, we start with a class specification file.

```
# who_wc.spec
#
# who word count DSI spec file
#
```

```
CLASS who_metrics = 150
LABEL "who wc data"
INDEX BY hour
MAX INDEXES 120
ROLL BY hour
RECORDS PER HOUR 60
;
```

```
METRICSwho_wc = 151
label "who wc"averaged
maximum 1000
precision 0
;
```

Compile the specification file to create a log file:

```
sdlcomp ./who_wc.spec ./who_wc_log.
```

Unlike `sar`, you cannot specify an interval or iteration value with `who`, so we create a script that provides, at a minimum, interval control.

```
#!/bin/ksh who_data_feed
while :
do
```

```
# sleep for one minute (this should correspond with the
# RECORDS PER HOUR clause in the specification file).
sleep 60

# Pipe the output of who into wc to count
# the number of users on the system.

who | wc -l > /usr/tmp/who_data
# copy the data record to the pipe being read by dsilog.

cat /usr/tmp/who_data > ./who.fifo
done
```

Again we need a fifo and a script to supply the data to **dsilog**, so we return to the step by step process.

1. Create two fifos; one will be the dummy fifo used to “hold open” the real input fifo
.# Dummy fifo.
mkfifo ./hold_open.fifo

Real input fifo for dsilog.
mkfifo ./who.fifo
2. Start **dsilog** using the -i option to specify the input coming from a fifo. It is important to start **dsilog** before starting the `who` data feed.
dsilog ./who_wc_log who_metrics \-i ./who.fifo &
3. Start the dummy process to hold open the input fifo.
cat ./hold_open.fifo \> ./who.fifo &
4. Start the `who` data feed script (`who_data_feed`).
./who_data_feed &

Chapter 18: Error Message

There are three types of DSI error messages: class specification, `dsilog` logging process, and general.

- Class specification error messages format consists of the prefix `SDL`, followed by the message number.
- `dsilog` logging process messages format consists of the prefix `DSILOG`, followed by the message number.
- General error messages can be generated by either of the above as well as other tasks. These messages have a minus sign (-) prefix and the message number.

DSI error messages are listed in this chapter. `SDL` and `DSILOG` error messages are listed in numeric order, along with the actions you take to recover from the error condition. General error messages are self-explanatory, so no recovery actions are given.

SDL Error Messages

SDL error messages are Self Describing Logfile class specification error messages, with the format, `SDL<message number>`.

DSILOG Error Messages

DSILOG error messages are `dsilog` logging process messages with the format,

`DSILOG<message number>`.

General Error Messages

Error	Explanation
-3	Attempt was made to add more classes than allowed by <code>max-class</code> .
-5	Could not open file containing class data.
-6	Could not read file.
-7	Could not write to file.
-9	Attempt was made to write to log file when write access was not requested.
-11	Could not find the pointer to the class.
-13	File or data structure not initialized.

Error	Explanation
-14	Class description file could not be read.
-15	Class description file could not be written to.
-16	Not all metrics needed to define a class were found in the metric description class.
-17	The path name of a file in the log file set is more than 1024 characters long.
-18	Class name is more then 20 characters long.
-19	File is not log file set root file.
-20	File is not part of a lod file set.
-21	The current software cannot access the log file set.
-22	Could not get shared memory segment or id.
-23	Could not attach to shared memory segment.
-24	Unable to open log file set.
-25	Could not determine current working directory.
-26	Could not read class header from class data file.
-27	Open of file in log file set failed.
-28	Could not open data class.
-29	Lseek failed.
-30	Could not read from log file.
-31	Could not write on log file.
-32	Remove failed.
-33	shmctl (REM_ID) failed.
-34	Log file set is incomplete: root or description file is missing.
-35	The target log file for adding a class is not in the current log file set.

Chapter 19: What is Transaction Tracking?

This chapter describes:

- [Improving Performance Management](#)
- [A Scenario: Real Time Order Processing](#)
- [Monitoring Transaction Data](#)

Improving Performance Management

You can improve your ability to manage system performance with the transaction tracking capability of HP Operations Agent and HP GlancePlus.

As the number of distributed mission-critical business applications increases, application and system managers need more information to tell them how their distributed information technology (IT) is performing.

- Has your application stopped responding?
- Is the application response time unacceptable?
- Are your service level objectives (SLOs) being met?

The transaction tracking capabilities of Performance Collection Component and GlancePlus allow IT managers to build in end-to-end manageability of their client/server IT environment in business transaction terms. With Performance Collection Component, you can define what a business transaction is and capture transaction data that makes sense in the context of *your* business.

When your applications are instrumented with the standard Application Response Measurement (ARM) API calls, these products provide extensive transaction tracking and end-to-end management capabilities across multi-vendor platforms.

Benefits of Transaction Tracking

- Provides a client view of elapsed time from the beginning to the end of a transaction.
- Provides transaction data.
- Helps you manage service level agreements (SLAs).

These topics are discussed in more detail in the remainder of this section.

Client View of Transaction Times

Transaction tracking provides you with a client view of elapsed time from the beginning to the end of a transaction. When you use transaction tracking in your Information Technology (IT) environment, you see the following benefits:

- You can accurately track the number of times each transaction executes.
- You can see how long it takes for a transaction to complete, rather than approximating the time as happens now.
- You can correlate transaction times with system resource utilization.
- You can use your own business deliverable production data in system management applications, such as data used for capacity planning, performance management, accounting, and charge-back.
- You can accomplish application optimization and detailed performance troubleshooting based on a real unit of work (your transaction), rather than representing actual work with abstract definitions of system and network resources.

Transaction Data

When Application Response Measurement (ARM) API calls have been inserted in an application to mark the beginning and end of each business transaction, you can then use the following resource and performance monitoring tools to monitor transaction data:

- Performance Collection Component provides the registration functionality needed to log, report, and detect alarms on transaction data. Transaction data can be viewed in Performance Manager, Glance, or by exporting the data from Performance Collection Component log files into files that can be accessed by spreadsheet and other reporting tools.
- Performance Manager graphs performance data for short-term troubleshooting and for examining trends and long-term analysis.
- Glance displays detailed real time data for monitoring your systems and transactions moment by moment.
- Performance Manager, Glance, or the HP Operations Manager message browser allow you to monitor alarms on service level compliance.

Individual transaction metrics are described in [Transaction Metrics](#).

Service Level Objectives

Service level objectives (SLOs) are derived from the stated service levels required by business application users. SLOs are typically based on the development of the service level agreement (SLA). From SLOs come the actual metrics that Information Technology resource managers need to collect, monitor, store, and report on to determine if they are meeting the agreed upon service levels for the business application user.

An SLO can be as simple as monitoring the response time of a simple transaction or as complex as tracking system availability.

A Scenario: Real Time Order Processing

Imagine a successful television shopping channel that employs hundreds of telephone operators who take orders from viewers for various types of merchandise. Assume that this enterprise uses a computer program to enter the order information, check merchandise availability, and update the stock

inventory. We can use this fictitious enterprise to illustrate how transaction tracking can help an organization meet customer commitments and SLOs.

Based upon the critical tasks, the customer satisfaction factor, the productivity factor, and the maximum response time, resource managers can determine the level of service they want to provide to their customers.

[Transaction Tracking Examples](#) contains a pseudocode example of how ARM API calls can be inserted in a sample order processing application so that transaction data can be monitored with Performance Collection Component and Glance.

Requirements for Real Time Order Processing

To meet SLOs in the real time order processing example described above, resource managers must keep track of the length of time required to complete the following critical tasks:

- Enter order information
- Query merchandise availability
- Update stock inventory

The key customer satisfaction factor for customers is how quickly the operators can take their order.

The key productivity factor for the enterprise is the number of orders that operators can complete each hour.

To meet the customer satisfaction and productivity factors, the response times of the transactions that access the inventory database, adjust the inventory, and write the record back must be monitored for compliance to established SLOs. For example, resource managers may have established an SLO for this application that 90 percent of the transactions must be completed in five seconds or less.

Preparing the Order Processing Application

ARM API calls can be inserted into the order processing application to create transactions for `inventory response` and `update inventory`. Note that the ARM API calls must be inserted by application programmers *prior* to compiling the application. See [Transaction Tracking Examples](#) for an example order processing program (written in pseudocode) that includes ARM API calls that define various transactions.

For more information on instrumenting applications with ARM API calls, see the [Application Response Measurement 2.0 API Guide](#).

Monitoring Transaction Data

When an application that is instrumented with ARM API calls is installed and running on your system, you can monitor transaction data with Performance Collection Component, GlancePlus, or Performance Manager.

... **with Performance Collection Component**

Using Performance Collection Component, you can collect and log data for named transactions, monitor trends in your SLOs over time, and generate alarms when SLOs are exceeded. Once these trends have been identified, Information Technology costs can be allocated based on transaction volumes. Performance Collection Component alarms can be configured to activate a technician's pager, so that problems can be investigated and resolved immediately. For more information, see [Advanced Features](#).

Performance Collection Component is required for transaction data to be viewed in Performance Manager.

... with Performance Manager

Performance Manager receives alarms and transaction data from Performance Collection Component. For example, you can configure Performance Collection Component so that when an order processing application takes too long to check stock, Performance Manager receives an alarm and sends a warning to the resource manager's console as an alert of potential trouble.

In Performance Manager, you can select **TRANSACTION** from the Class List window for a data source, then **graph transaction metrics** for various transactions. For more information, see Performance Manager online help.

... with GlancePlus

Use GlancePlus to monitor up-to-the-second transaction response time and whether or not your transactions are performing within your established SLOs. GlancePlus helps you identify and resolve resource bottlenecks that may be impacting transaction performance. For more information, see the GlancePlus online help, which is accessible through the GlancePlus Help menu.

Guidelines for Using ARM

Instrumenting applications with the ARM API requires some careful planning. In addition, managing the environment that has ARMed applications in it is easier if the features and limitations of ARM data collection are understood. Here is a list of areas that could cause some confusion if they are not fully understood.

1. In order to capture ARM metrics, `ttd` and `midaemon` must be running. For Performance Collection Component, the `scope` collector must be running to log ARM metrics. The `ovpa start` script starts all required processes. Likewise, Glance starts `ttd` and `midaemon` if they are not already active. (See [Transaction Tracking Daemon \(ttd\) in Chapter 19](#))
2. Re-read the transaction configuration file, `ttd.conf`, to capture any newly-defined transaction names. (See [Transaction Configuration File \(ttd.conf\) in Chapter 19](#))
3. Performance Collection Component, user applications, and `ttd` must be restarted to capture any *new* or modified transaction ranges and service level objectives (SLOs). (See [Adding New Applications in Chapter 19](#))
4. Strings in user-defined metrics are ignored by Performance Collection Component. Only the first six non-string user-defined metrics are logged. (See [How Data Types Are Used in Chapter 24](#))
5. Using dashes in the transaction name has limitations if you are specifying an alarm condition for that transaction. (See "... with Performance Collection Component" in the section [Alarms in Chapter 20](#))

6. Performance Collection Component will only show the first 60 characters in the application name and transaction name. (See [Specifying Application and Transaction Names in Chapter 19](#))
7. Limit the number of unique transaction names that are instrumented. (See [Limits on Unique Transaction in Chapter 20](#))
8. Do not allow ARM API function calls to affect the execution of an application from an end-user perspective. (See [ARM API Call Status Returns in Chapter 19](#))
9. Use shared libraries for linking. (See the section "[C Compiler Option Examples by Platform](#)" on [page 372](#))

Chapter 20: How Transaction Tracking Works

The following components of Performance Collection Component and GlancePlus work together to help you define and track transaction data from applications instrumented with Application Response Measurement (ARM) calls.

- The Measurement Interface daemon, `midaemon`, is a daemon process that monitors and reports transaction data to its shared memory segment where the information can be accessed and reported by Performance Collection Component, Performance Manager, and GlancePlus. On HP-UX systems, the `midaemon` also monitors system performance data.
- The transaction configuration file, `/var/opt/perf/ttd.conf`, is used to define transactions and identify the information to monitor for each transaction.
- The Transaction Tracking daemon, `ttd`, reads, registers, and synchronizes transaction definitions from the transaction configuration file, `ttd.conf`, with the `midaemon`.

Support of ARM 2.0

ARM 2.0 is a superset of the previous version of Application Response Measurement. The new features that ARM 2.0 provides are user-defined metrics, transaction correlation, and a logging agent. Performance Collection Component and GlancePlus support user-defined metrics and transaction correlation but *do not* support the logging agent.

However, you may want to use the logging agent to test the instrumentation in your application. The source code for the logging agent, `logagent.c`, is included in the ARM 2.0 Software Developers Kit (SDK) that is available from the following web site:

<http://regions.cmg.org/regions/cmgarmlw>

For information about using the logging agent, see the [Application Response Measurement 2.0 API Guide](#).

Note: The *Application Response Measurement 2.0 API Guide* uses the term “application-defined metrics” instead of “user-defined metrics”.

Support of ARM API Calls

The Application Response Measurement (ARM) API calls listed below are supported in Performance Collection Component and GlancePlus.

<code>arm_init()</code>	Names and registers the application and (optionally) the user.
-------------------------	--

<code>arm_getid()</code>	Names and registers a transaction class, and provides related transaction information. Defines the context for user-defined metrics.
<code>arm_start()</code>	Signals the start of a unique transaction instance.
<code>arm_update()</code>	Updates the values of a unique transaction instance.
<code>arm_stop()</code>	Signals the end of a unique transaction instance.
<code>arm_end()</code>	Signals the end of the application.

See your current [Application Response Measurement 2.0 API Guide](#) and the *arm* (3) man page for information on instrumenting applications with ARM API calls as well as complete descriptions of the calls and their parameters. For commercial applications, check the product documentation to see if the application has been instrumented with ARM API calls.

For important information about required libraries, see the [Transaction Libraries](#) later in this manual.

arm_complete_transaction Call

In addition to the ARM 2.0 API standard, the HP ARM agent supports the `arm_complete_transaction` call. This call, which is an HP-specific extension to the ARM standard, can be used to mark the end of a transaction that has completed when the start of the transaction could not be delimited by an `arm_start` call. The `arm_complete_transaction` call takes as a parameter the response time of the completed transaction instance.

In addition to signaling the end of a transaction instance, additional information about the transaction can be provided in the optional data buffer. See the *arm* (3) man page for more information on this optional data as well a complete description of this call and its parameters.

Sample ARM-Instrumented Applications

For examples of how ARM API calls are implemented, see the sample ARM-instrumented applications, `armsample1.c`, `armsample2.c`, `armsample3.c`, and `armsample4.c`, and their build script, `Make.armsample`, in the `<InstallDir>/examples/arm/` directory.

- `armsample1.c` shows the use of simple standard ARM API calls.
- `armsample2.c` also shows the use of simple standard ARM API calls. It is similar in structure to `armsample1.c`, but is interactive.
- `armsample3.c` provides examples of how to use the user-defined metrics and the transaction correlator, provided by version 2.0 of the ARM API. This example simulates a client/server application where both server and client perform a number of transactions. (Normally application client and server components would exist in separate programs, but they are put together for simplicity.)

The client procedure starts a transaction and requests an ARM correlator from its `arm_start` call. This correlator is saved by the client and passed to the server so that the server can use it when it

calls `arm_start`. The performance tools running on the server can then use this correlator information to distinguish the different clients making use of the server.

Also shown in this program is the mechanism for passing user-defined metric values into the ARM API. This allows you to not only see the response times and service-level information in the performance tools, but also data which may be important to the application itself. For example, a transaction may be processing different size requests, and the size of the request could be a user-defined metric. When the response times are high, this user-defined metric could be used to see if long response times correspond to bigger size transaction instances.

- `armsample4.c` provides an example of using user-defined metrics in ARM calls. Different metric values can be passed through `arm_start`, `arm_update`, and `arm_stop` calls. Alternatively, `arm_complete_transaction` can be used where a tran cannot be delimited by `start/stop` calls.

Specifying Application and Transaction Names

Although ARM allows a maximum of 128 characters each for application and transaction names in the `arm_init` and `arm_getid` API calls, Performance Collection Component shows *only* a maximum of 60 characters. All characters beyond the first 60 will not be visible. However, GlancePlus allows you to view up to 128 characters.

Performance Collection Component applies certain limitations to how application and transaction names are shown in extracted or exported transaction data. These rules also apply to viewing application and transaction names in Performance Manager.

The application name *always* takes precedence over the transaction name. For example, if you are exporting transaction data that has a 65-character application name and a 40-character transaction name, *only* the application name is shown. The last five characters of the application name are not visible.

For another example, if an application name contains 32 characters and the transaction name has 40 characters, Performance Collection Component shows the entire application name but the transaction name appears truncated. A total of 60 characters are shown. Fifty-nine characters are allocated to the application and transaction names and one character is allocated to the underscore (`_`) that separates the two names. This is how the application name “WarehouseInventoryApplication” and the transaction name “CallFromWestCoastElectronicSupplier” would appear in Performance Collection Component or Performance Manager:

```
WarehouseInventoryApplication_CallFromWestCoastElectronicSup
```

Note: The 60-character combination of application name and transaction name must be unique if the data is to be viewed with Performance Manager.

Transaction Tracking Daemon (ttd)

The Transaction Tracking daemon, `ttd`, reads, registers, and synchronizes transaction definitions from `ttd.conf` with `midaemon`.

ttd is started when you start up Performance Collection Component's scope data collector with the `ovpa start` command. ttd runs in background mode when dispatched, and errors are written to the file `/var/opt/perf/status.ttd`.

midaemon must also be running to process the transactions and to collect performance metrics associated with these transactions (see next page).

Caution: We strongly recommend that you do not stop ttd.

If you must stop ttd, any ARM-instrumented applications that are running *must* also be stopped before you restart ttd and Performance Collection Component processes. ttd must be running to capture all `arm_init` and `arm_getid` calls being made on the system. If ttd is stopped and restarted, transaction IDs returned by these calls will be repeated, thus invalidating the ARM metrics

Use the `ovpa` script to start Performance Collection Component processes to ensure that the processes are started in the correct order. `ovpa stop` will *not* shut down ttd. If ttd must be shut down for a reinstall of any performance software, use the command `<InstallDir>/bin/ttd -k`. However, we do not recommend that you stop ttd, except when reinstalling Performance Collection Component.

If Performance Collection Component is not on your system, GlancePlus starts `midaemon`. `midaemon` then starts ttd if it is not running *before* `midaemon` starts processing any measurement data.

See the ttd man page for complete program options.

ARM API Call Status Returns

The ttd process must always be running in order to register transactions. If ttd is killed for any reason, while it is not running, `arm_init` or `arm_getid` calls will return a "failed" return code. If ttd is subsequently restarted, new `arm_getid` calls may re-register the same transaction IDs that are already being used by other programs, thus causing invalid data to be recorded.

When ttd is killed and restarted, ARM-instrumented applications may start getting a return value of `-2` (`TT_TTDNOTRUNNING`) and an `EPIPE` error on ARM API calls. When your application initially starts, a client connection handle is created on any initial ARM API calls. This client handle allows your application to communicate with the ttd process. When ttd is killed, this connection is no longer valid and the next time your application attempts to use an ARM API call, you may get a return value of `TT_TTDNOTRUNNING`. This error reflects that the *previous* ttd process is no longer running even though there is another ttd process running. (Some of the ARM API call returns are explained in the `arm (3)` man page.)

To get around this error, you must restart your ARM-instrumented applications if ttd is killed. First, stop your ARMed applications. Next, restart ttd (using `<InstallDir>/bin/ovpa start` or `<InstallDir>/bin/ttd`), and then restart your applications. The restart of your application causes the creation of a new client connection handle between your application and the ttd process.

Some ARM API calls will not return an error if the `midaemon` has an error. For example, this would occur if the `midaemon` has run out of room in its shared memory segment. The performance metric `GBL_TT_OVERFLOW_COUNT` will be `> 0`. If an overflow condition occurs, you may want to shut down any performance tools that are running (except ttd) and restart the `midaemon` using the `-smdvss` option to specify more room in the shared memory segment. (For more information, see the `midaemon` man page.)

We recommend that your applications be written so that they continue to execute even if ARM errors occur. ARM status should not affect program execution.

The number of active client processes that can register transactions with `ttd` via the `arm_getid` call is limited to the `maxfiles` kernel parameter. This parameter controls the number of open files per process. Each client registration request results in `ttd` opening a socket (an open file) for the RPC connection. The socket is closed when the client application terminates. Therefore, this limit affects only the number of active clients that have registered a transaction via the `arm_getid` call. Once this limit is reached, `ttd` will return `TT_TTDNOTRUNNING` to a client's `arm_getid` request. The `maxfiles` kernel parameter can be increased to raise this limit above the number of active applications that will register transactions with `ttd`.

Measurement Interface Daemon (midaemon)

The Measurement Interface daemon, `midaemon`, is a low-overhead process that continuously collects system performance information. The `midaemon` must be running for Performance Collection Component to collect transaction data or for GlancePlus to report transaction data. It starts running when you run the `scope` or `perfd` process or when starting GlancePlus.

Performance Collection Component and GlancePlus require both the `midaemon` and `ttd` to be running so that transactions can be registered and tracked. The `ovpa` script starts and stops Performance Collection Component processing, including the `midaemon`, in the correct order. GlancePlus starts the `midaemon`, if it is not already running. The `midaemon` starts `ttd`, if it is not already running.

See the "[CPU Overhead](#)" on page 351 section later in this manual for information on the `midaemon` CPU overhead.

See the `midaemon` man page for complete program options.

Transaction Configuration File (ttd.conf)

The transaction configuration file, `/var/opt/perf/ttd.conf`, allows you to define the application name, transaction name, the performance distribution ranges, and the service level objective you want to meet for each transaction. The `ttd` reads `ttd.conf` to determine how to register each transaction.

Customization of `ttd.conf` is optional. The default configuration file that ships with Performance Collection Component causes *all* transactions instrumented in any application to be monitored.

If you are using a commercial application and don't know which transactions have been instrumented in the application, collect some data using the default `ttd.conf` file. Then look at the data to see which transactions are available. You can then customize the transaction data collection for that application by modifying `ttd.conf`.

Adding New Applications

If you add new ARMED applications to your system that use the default `slo` and `range` values from the `tran=*` line in your `ttd.conf` file, you don't need to do anything to incorporate these new transactions. (See the [Configuration File Keywords](#) section for descriptions of `tran`, `range`, and `slo`.) The new

transactions will be picked up automatically. The `slo` and `range` values from the `tran` line in your `ttd.conf` file will be applied to the new transactions.

Adding New Transactions

After making additions to the `ttd.conf` file, you must perform the following steps to make the additions effective:

- Stop all applications.
- Execute the `ttd -hup -mi` command as root.

The above actions cause the `ttd.conf` file to be re-read and registers the new transactions, along with their `slo` and `range` values, with `ttd` and the `midaemon`. The re-read will not change the `slo` or `range` values for any transactions that were in the `ttd.conf` file prior to the re-read.

Changing the Range or SLO Values

If you need to change the SLO or range values of existing transactions in the `ttd.conf` file, you must do the following:

- Stop all ARMed applications.
- Stop the scope collector using `ovpa stop`.
- Stop any usage of Glance.
- Stop the `ttd` by issuing the command `ttd -k`.
- Once you have made your changes to the `ttd.conf` file:
- Restart scope using `ovpa start`.
- Restart your ARMed applications.

Configuration File Keywords

The `/var/opt/perf/ttd.conf` configuration file associates transaction names with transaction attributes that are defined by the keywords in Table 1.

Table 1: Configuration File Keywords

Keyword	Syntax	Usage
<code>tran</code>	<code>tran=<i>transaction_name</i></code>	Required
<code>slo</code>	<code>slo=<i>sec</i></code>	Optional
<code>range</code>	<code>range=<i>sec</i> [<i>,sec,...</i>]</code>	Optional

These keywords are described in more detail below.

tran

Use `tran` to define your transaction name. This name must correspond to a transaction that is defined in the `arm_getid` API call in your instrumented application. You must use the `tran` keyword before you can specify the optional attributes `range` or `slo`. `tran` is the only required keyword within the configuration file. A trailing asterisk (*) in the transaction name causes a wild card pattern match to be performed when registration requests are made against this entry. Dashes can be used in a transaction name. However, spaces cannot be used in a transaction name.

The transaction name can contain a maximum of 128 characters. However, only the first 60 characters are visible in Performance Collection Component. GlancePlus can display 128 characters in specific screens.

The default `ttd.conf` file contains several entries. The first entries define transactions used by the Performance Collection Component data collector `scope`, which is instrumented with ARM API calls. The file also contains the entry `tran=*`, which registers all other transactions in applications instrumented with ARM API or Transaction Tracker API calls.

range

Use `range` to specify the transaction performance distribution ranges. Performance distribution ranges allow you to distinguish between transactions that take different lengths of time to complete and to see how many successful transactions of each length occurred. The ranges that you define appear in the GlancePlus Transaction Tracking window.

Each value entered for `sec` represents the upper limit in seconds for the transaction time for the range. The value may be an integer or real number with a maximum of six digits to the right of the decimal point. On HP-UX, this allows for a precision of one microsecond (.000001 seconds). On other platforms, however, the precision is ten milliseconds (0.01 seconds), so only the first two digits to the right of the decimal point are recognized.

A maximum of ten ranges are supported for each transaction you define.

You can specify up to nine ranges. One range is reserved for an overflow range, which collects data for transactions that take longer than the largest user-defined range. If you specify more than nine ranges, the first nine ranges are used and the others are ignored.

If you specify fewer than nine ranges, the first unspecified range becomes the overflow range. Any remaining unspecified ranges are not used. The unspecified range metrics are reported as 0.000. The first corresponding unspecified count metric becomes the overflow count. Remaining unspecified count metrics are always zero (0).

Ranges must be defined in ascending order (see examples later in this chapter).

slo

Use `slo` to specify the service level objective (SLO) in seconds that you want to use to monitor your performance service level agreement (SLA).

As with the `range` keyword, the value may be an integer or real number, with a maximum of six digits to the right of the decimal point. On HP-UX, this allows for a precision of one microsecond (.000001

seconds). On other platforms, however, the precision is ten milliseconds (0.01 seconds), so only the first two digits to the right of the decimal point are recognized.

Note that even though transactions can be sorted with one microsecond precision on HP-UX, transaction times are displayed with 100 microsecond precision.

Configuration File Format

The `ttd.conf` file can contain two types of entries: general transactions and application-specific transactions.

General transactions should be defined in the `ttd.conf` file before any application is defined. These transactions will be associated with all the applications that are defined. The default `ttd.conf` file contains one general transaction entry and entries for the `scope` collector that is instrumented with ARM API calls.

```
tran=* range=0.5, 1, 2, 3, 5, 10, 30, 120, 300 slo=5.0
```

Optionally, each application can have its own set of transaction names. These transactions will be associated *only* with that application. The application name you specify must correspond to an application name defined in the `arm_init` API call in your instrumented application. Each group of application-specific entries must begin with the name of the application enclosed in brackets. For example:

```
[AccountRec]
```

```
tran=acctOne range=0.01, 0.03, 0.05
```

The application name can contain a maximum of 128 characters. However, only the first 60 characters are visible in Performance Collection Component. Glance can display 128 characters in specific screens.

If there are transactions that have the same name as a “general” transaction, the transaction listed under the application will be used.

For example:

```
tran=abc range=0.01, 0.03, 0.05 slo=0.10
```

```
tran=xyz range=0.02, 0.04, 0.06 slo=0.08
```

```
tran=t* range=0.01, 0.02, 0.03
```

```
[AccountRec}
```

```
tran=acctOne range=0.04, 0.06, 0.08
```

```
tran=acctTwo range=0.1, 0.2
```

```
tran=t* range=0.03, 0.5
```

```
[AccountPay]
```

```
[GenLedg]
```

```
tran=GenLedgOne range=0.01
```

In the example above, the first three transactions apply to all of the three applications specified.

The application [AccountRec] has the following transactions: acctOne, acctTwo, abc, xyz, and t*. One of the entries in the general transaction set also has a wild card transaction named "t*". In this case, the "t*" transaction name for the AccountRec application will be used; the one in the general transaction set is ignored.

The application [AccountPay] has only transactions from the general transactions set.

The application [GenLedg] has transactions GenLedgOne, abc, xyz, and t*.

The ordering of transactions names makes no difference within the application.

For additional information about application and transaction names, see the section ["Specifying Application and Transaction Names" on page 343](#) in this chapter.

Configuration File Examples

Example 1

```
tran=* range=0.5,1,2,3,5,10,30,12,30 slo=5.0
```

The "*" entry is used as the default if none of the entries match a registered transaction name. These defaults can be changed on each system by modifying the "*" entry. If the "*" entry is missing, a default set of registration parameters are used that match the initial parameters assigned to the "*" entry above.

Example 2

```
[MANufactr]
tran=MFG01 range=1,2,3,4,5,10 slo=3.0
tran=MFG02 range=1,2.2,3.3,4.0,5.5,10 slo=4.5
tran=MFG03
tran=MFG04 range=1,2.2,3.3,4.0,5.5,10
```

Transactions for the MANufactr application, MFG01, MFG02, and MFG04, each use their own unique parameters. The MFG03 transaction does not need to track time distributions or service level objectives so it does not specify these parameters.

Example 3

```
[Financial]
tran=FIN01
tran=FIN02 range=0.1,0.5,1,2,3,4,5,10,20 slo=1.0
tran=FIN03 range=0.1,0.5,1,2,3,4,5,10,20 slo=2.0
```

Transactions for the Financial application, FIN02 and FIN03, each use their own unique parameters. The FIN01 transaction does not need to track time distributions or service level objectives so it does not specify these parameters.

Example 4

```
[PERSONL]
```

```
tran=PERS* range=0.1,0.5,1,2,3,4,5,10,20 slo=1.0
```

```
tran=PERS03 range=0.1,0.2,0.5,1,2,3,4,5,10,20 slo=0.8
```

The PERS03 transaction for the PERSONL application uses its own unique parameters while the remainder of the personnel transactions use a default set of parameters unique to the PERSONL application.

Example 5

```
[ACCOUNTS]
```

```
tran=ACCT_* slo=1.0
```

```
tran=ACCT_REC range=0.5,1,2,3,4,5,10,20 slo=2.0
```

```
tran=ACCT_PAY range=0.5,1,2,3,4,5,10,20 slo=2.0
```

Transactions for the ACCOUNTS application, ACCT_REC and ACCT_PAY, each use their own unique parameters while the remainder of the accounting transactions use a default set of parameters unique to the accounting application. Only the accounts payable and receivable transactions need to track time distributions. The order of transaction names makes no difference within the application.

Overhead Considerations for Using ARM

The current versions of Performance Collection Component and GlancePlus contain modifications to their measurement interface that support additional data required for ARM 2.0. These modifications can result in increased overhead for performance management. You should be aware of overhead considerations when planning ARM instrumentation for your applications.

The overhead areas are discussed in the remainder of this chapter.

Guidelines

Here are some guidelines to follow when instrumenting your applications with the ARM API:

- The total number of separate transaction IDs should be limited to not more than 4,000. Generally, it is cheaper to have multiple instances of the same transaction than it is to have single instances of different transactions. Register *only* those transactions that will be actively monitored.
- Although the overhead for the `arm_start` and `arm_stop` API calls is very small, it can increase when there is a large volume of transaction instances. More than a few thousand `arm_start` and `arm_stop` calls per second on most systems can significantly impact overall performance.
- Request ARM correlators *only* when using ARM 2.0 functionality. (For more information about ARM correlators, see the “Advanced Topics” section in the [Application Response Measurement 2.0 API Guide](#). The overhead for producing, moving, and monitoring correlator information is significantly higher than for monitoring transactions that are not instrumented to use the ARM 2.0 correlator functionality.
- Larger string sizes (applications registering lengthy transaction names, application names, and user-defined string metrics) will impose additional overhead.

Disk I/O Overhead

The performance management software does not impose a large disk overhead on the system. Glance generally does not log its data to disk. Performance Collection Component's collector daemon, `scope`, generates disk log files, but their size is not significantly impacted by ARM 2.0. The `logtran scope` log file is used to store ARM data.

CPU Overhead

A program instrumented with ARM calls will generally not run slower because of the ARM calls. This assumes that the rate of `arm_getid` calls is lower than one call per second, and the rate of `arm_start` and `arm_stop` calls is lower than a few thousand per second. More frequent calls to the ARM API should be avoided.

Most of the additional CPU overhead for supporting ARM is incurred inside of the performance tool programs and daemons themselves. The `midaemon` CPU overhead rises slightly but not more than two percent more than it was with ARM 1.0. If the `midaemon` has been requested to track per-transaction resource metrics, the overhead per transaction instance may be twice as high as it would be without tracking per-transaction resource metrics. (You can enable the tracking of per-transaction resource metrics by setting the `log transaction=resource` flag in the `parm` file.) In addition, Glance and `scope` CPU overhead will be slightly higher on a system with applications instrumented with ARM 2.0 calls. Only those applications that are instrumented with ARM 2.0 calls that make extensive use of correlators and/or user-defined metrics will have a significant performance impact on the `midaemon`, `scope`, or Glance.

An `midaemon` overflow condition can occur when usage exceeds the available default shared memory. This results in:

- No return codes from the ARM calls once the overflow condition occurs.
- Display of incorrect metrics, including blank process names.
- Errors being logged in `status.mi` (for example, "out of space").

Memory Overhead

Programs that are making ARM API calls will not have a significant impact in their memory virtual set size, except for the space used to pass ARM 2.0 correlator and user-defined metric information. These buffers, which are explained in the [Application Response Measurement 2.0 API Guide](#), should not be a significant portion of a process's memory requirements.

There is additional virtual set size overhead in the performance tools to support ARM 2.0. The `midaemon` process creates a shared memory segment where ARM data is kept internally for use by Performance Collection Component and GlancePlus. The size of this shared memory segment has grown, relative to the size on releases with ARM 1.0, to accommodate the potential for use by ARM 2.0. By default on most systems, this shared memory segment is approximately 11 megabytes in size. This segment is not all resident in physical memory unless it is required. Therefore, this should not be a significant impact on most systems that are not already memory-constrained. The memory overhead of `midaemon` can be tuned using special startup parameters (see the `midaemon` man page).

Chapter 21: Getting Started with Transactions

This chapter gives you the information you need to begin tracking transactions and your service level objectives. For detailed reference information, see ["How Transaction Tracking Works" on page 341](#). See ["Transaction Tracking Examples" on page 361](#) for examples.

Before you start

Performance Collection Component provides the `libarm.*` shared library in the following locations:

Platform	Path
IBM RS/6000	<code>/usr/lpp/perf/lib/</code>
Other UNIX platforms	<code>/opt/perf/lib/</code>

If you do not have Performance Collection Component installed on your system and if `libarm.*` doesn't exist in the path indicated above for your platform, see ["C Compiler Option Examples by Platform" on page 372](#) at the end of this manual. See also "The ARM Shared Library (`libarm`)" section in the [Application Response Measurement 2.0 API Guide](#) for information on how to obtain it. For a description of `libarm`, see ["ARM Library \(`libarm`\)" on page 369](#) at the end of this manual.

Setting Up Transaction Tracking

Follow the procedure below to set up transaction tracking for your application. These steps are described in more detail in the remainder of this section.

1. Define SLOs by determining what key transactions you want to monitor and the response level you expect (*optional*).
2. To monitor transactions in Performance Collection Component and Performance Manager, make sure that the Performance Collection Component `parm` file has transaction logging turned on. Then start or restart Performance Collection Component to read the updated `parm` file.

Editing the `parm` file is *not* required to see transactions in GlancePlus. However, `ttd` *must* be running in order to see transactions in GlancePlus. Starting GlancePlus will automatically start `ttd`.

3. Run the application that has been instrumented with ARM API calls that are described in this manual and the [Application Response Measurement 2.0 API Guide](#).

4. Use Performance Collection Component or Performance Manager to look at the collected transaction data, or use GlancePlus to view current data. If the data isn't visible in Performance Manager, close the data source and then reconnect to it.
5. Customize the configuration file, `ttd.conf`, to modify the way transaction data for the application is collected (*optional*).
6. After making additions to the `ttd.conf` file, you must perform the following steps to make the additions effective:
 - a. Stop all ARMed applications.
 - b. Execute the `ttd -hup -mi` command as `root`.

These actions re-read the `ttd.conf` file and registers new transactions along with their `slo` and `range` values with `ttd` and the `midaemon`. The re-read will not change the `slo` or `range` values for any transactions that were in the `ttd.conf` file prior to the re-read.

7. If you need to change the `slo` or `range` values of existing transactions in the `ttd.conf` file, do the following:
 - a. Stop all ARMed applications.
 - b. Stop the `scope` collector using `ovpa stop`.
 - c. Stop all usage of Glance.
 - d. Stop `ttd` using `ttd -k`.

Once you have made your changes:

- a. Restart `scope` using `ovpa start`.
- b. Start your ARMed applications.

Defining Service Level Objectives

Your first step in implementing transaction tracking is to determine the key transactions that are required to meet customer expectations and what level of transaction responsiveness is required. The level of responsiveness that is required becomes your service level objective (SLO). You define the service level objective in the configuration file, `ttd.conf`.

Defining service level objectives can be as simple as reviewing your Information Technology department's service level agreement (SLA) to see what transactions you need to monitor to meet your SLA. If you don't have an SLA, you may want to implement one. However, creating an SLA is not required in order to track transactions.

Modifying the Parm File

If necessary, modify the Performance Collection Component `parm` file to add transactions to the list of items to be logged for use with Performance Manager and Performance Collection Component. Include the `transaction` option in the `parm` file's `log` parameter as shown in the following example:

```
log global application process transaction device=disk
```

The default for the `log transaction` parameter is `no resource` and `no correlator`. To turn on resource data collection or correlator data collection, specify `log transaction=resource` or `log transaction=correlator`. Both can be logged by specifying `log transaction=resource, correlator`.

Before you can collect transaction data for use with Performance Collection Component and Performance Manager, the updated `parm` file must be activated as described below:

Performance Collection Component status	Command to activate transaction tracking
Running	<code>ovpa restart</code>
Not running	<code>ovpa start</code>

Collecting Transaction Data

Start up your application. The Transaction Tracking daemon, `ttd`, and the Measurement Interface daemon, `midaemon`, collect and synchronize the transaction data for your application as it runs. The data is stored in the `midaemon`'s shared memory segment where it can be used by Performance Collection Component or GlancePlus. See ["Monitoring Performance Data" on page 357](#) for information on using each of these tools to view transaction data for your application.

Error Handling

Due to performance considerations, not all problematic ARM or Transaction Tracker API calls return errors in real time. Some examples of when errors are not returned as expected are:

- calling `arm_start` with a bad `id` parameter such as an uninitialized variable
- calling `arm_stop` without a previously successful `arm_start` call

Performance Collection Component — To debug these situations when instrumenting applications with ARM calls, run the application long enough to generate and collect a sufficient amount of transaction data. Collect this data with Performance Collection Component, then use the `extract` program's `export` command to export data from the `logtran` file. Examine the data to see if all transactions were logged as expected. Also, check the `/var/opt/perf/status.ttd` file for possible errors.

GlancePlus — To debug these situations when instrumenting applications with ARM calls, run the application long enough to generate a sufficient amount of transaction data, then use GlancePlus to see if all transactions appear as expected.

Limits on Unique Transactions

Depending on your particular system resources and kernel configuration, a limit may exist on the number of unique transactions allowed in your application. This limit is normally several thousand unique `arm_getid` calls.

The number of unique transactions may exceed the limit when the shared memory segment used by `midaemon` is full. If this happens, an overflow message appears in GlancePlus. Although no message appears in Performance Collection Component, data for subsequent new transactions won't be logged. (However, check `/var/opt/perf/status.scope` for an overflow message.) Data for subsequent new transactions won't be visible in GlancePlus. Transactions that have already been registered will continue to be logged and reported. The `GBL_TT_OVERFLOW_COUNT` metric in GlancePlus reports the number of new transactions that could not be measured.

This situation can be remedied by stopping and restarting the `midaemon` process using the `-smdvss` option to specify a larger shared memory segment size. The current shared memory segment size can be checked using the `midaemon -sizes` command. For more information on optimizing the `midaemon` for your system, see the `midaemon` man page.

Customizing the Configuration File (optional)

After viewing the transaction data from your application, you may want to customize the transaction configuration file, `/var/opt/perf/ttd.conf`, to modify the way transaction data for the application is collected. This is optional because the default configuration file, `ttd.conf`, will work with all transactions defined in the application. If you do decide to customize the `ttd.conf` file, complete this task on the same systems where you run your application. You must be logged on as `root` to modify `ttd.conf`.

See ["How Transaction Tracking Works" on page 341](#) for information on the configuration file keywords – `tran`, `range`, and `slo`. Some examples of how each keyword is used are shown below:

```
tran=Example:   tran=answerid
                tran=answerid*
                tran=*
range=Example:  range=2.5,4.2,5.0,10.009
slo=Example:    slo=4.2
```

Customize your configuration file to include all of your transactions and each associated attribute. Note that the use of the `range` or `slo` keyword must be preceded by the `tran` keyword. An example of a `ttd.conf` file is shown below.

```
tran=*
tran=my_first_transaction slo=5.5
[answerid]
tran=answerid1 range=2.5, 4.2, 5.0, 10.009 slo=4.2
[orderid]
tran=orderid1 range=1.0, 1.5, 2.0, 2.5, 3.0, 3.5, 4.0
```

If you need to make additions to the `ttd.conf` file:

- Stop all ARMed applications.
- Execute the `ttd -hup -mi` command as `root`.

The above actions re-read the `ttd.conf` file and registers new transactions along with their `slo` and `range` values with `ttd` and the `midaemon`. The re-read will not change the `slo` or `range` value for any transactions that were in the `ttd.conf` file prior to the re-read,

If you need to change the `slo` or `range` values of existing transactions in the `ttd.conf` file, do the following:

1. Stop all ARMed applications.
2. Stop the scope collector using `ovpa stop`.
3. Stop all usage of Glance.
4. Stop `ttd` using `ttd -k`.

Once you have made your changes:

1. Restart `scope` using `ovpa start`.
2. Start your ARMed applications.

Monitoring Performance Data

You can use the following resource and performance management products to monitor transaction data – Performance Collection Component, Performance Manager, and GlancePlus.

... with Performance Collection Component

By collecting and logging data for long periods of time, Performance Collection Component gives you the ability to analyze your system's performance over time and to perform detailed trend analysis. Data from Performance Collection Component can be viewed with Performance Manager Agent or exported for use with a variety of other performance monitoring, accounting, modeling, and planning tools.

With Performance Collection Component's `extract` program, data can be exported for use with spreadsheets and analysis programs. Data can also be extracted for archiving and analysis.

Performance Collection Component and `ttd` must be running in order to monitor transaction data in Performance Collection Component. Starting Performance Collection Component using the `ovpa` script ensures that the `ttd` and `midaemon` processes that are required to view transaction data in GlancePlus are started in the right order.

... with Performance Manager

Performance Manager imports Performance Collection Component data and gives you the ability to translate that data into a customized graphical or numerical format. Using Performance Manager, you can perform analysis of historical trends of transaction data and you can perform more accurate forecasting.

You can select **TRANSACTION** from the Class List window for a data source in Performance Manager, then graph transaction metrics for various transactions. For more information, see Performance Manager online help, which is accessible from the Performance Manager Help menu. If you don't see the transactions you expect in Performance Manager, close the current data source and then reconnect to it.

... with GlancePlus

Monitoring systems with GlancePlus helps you identify resource bottlenecks and provides immediate performance information about the computer system. GlancePlus has a Transaction Tracking window that displays information about all transactions that you have defined and a Transaction Graph window that displays specific information about a single transaction. For example, you can see how each transaction is performing against the SLO that you have defined. For more information about how to use GlancePlus, see the online help that is accessible from the Help menu.

Alarms

You can alarm on transaction data with the following resource and performance management products — Performance Collection Component, Performance Manager, and GlancePlus.

... with Performance Collection Component

In order to generate alarms with Performance Collection Component, you must define alarm conditions in its alarm definitions file, `alarmdef`. You can set up Performance Collection Component to notify you of an alarm condition in various ways, such as sending an email message or initiating a call to your pager.

To pass a syntax check for the `alarmdef` file, you must have data logged for that application name and transaction name in the log files, or have the names registered in the `ttt.conf` file.

There is a limitation when you define an alarm condition on a transaction that has a dash (–) in its name. To get around this limitation, use the `ALIAS` command in the `alarmdef` file to redefine the transaction name.

... with GlancePlus

You can configure the Adviser Syntax to alarm on transaction performance. For example, when an alarm condition is met, you can instruct GlancePlus to display information to `stdout`, execute a UNIX command (such as `mailx`), or turn the Alarm button on the main GlancePlus window yellow or red. For more information about alarms in GlancePlus, choose **On This Window** from the Help menu in the Edit Adviser Syntax window.

Chapter 22: Transaction Tracking Messages

The error codes listed in Table 2 are returned and can be used by the application developer when instrumenting an application with Application Response Measurement (ARM) or Transaction Tracker API calls:

Table 2: Error codes

Error Code	Errno Value	Meaning
-1	EINVAL	Invalid arguments
-2	EPIPE	ttd (registration daemon) not running
-3	ESRCH	Transaction name not found in the ttd.conf file
-4	EOPNOTSUPP	Operating system version not supported

When an application instrumented with ARM or Transaction Tracker API calls is running, return codes from any errors that occur will probably be from the Transaction Tracking daemon, ttd. The Measurement Interface daemon, midaemon, does not produce any error return codes.

If a midaemon error occurs, see the `/var/opt/perf/status.mi` file for more information.

Chapter 23: Transaction Metrics

The ARM agent provided as a shared component of both the GlancePlus and Performance Collection Component, produces many different transaction metrics. To see a complete list of the metrics and their descriptions:

- For installed GlancePlus metrics, use the GlancePlus online help or see the *GlancePlus for HP-UX Dictionary of Performance Metrics* located:

On UNIX/Linux under `/<InstallDir>/paperdocs/gp/C/` as `gp-metrics.txt`.

`InstallDir` is the directory in which Performance Collection Component is installed.

- For installed Performance Collection Component metrics for specific platforms, see the platform's *HP Operations Agent Dictionary of Operating System Performance Metrics* files located:

On UNIX/Linux under `/<InstallDir>/paperdocs/ovpa/C/` as `met<platform>.txt`.

On Windows under `%ovinstalldir%paperdocs\ovpa\C` as `met<platform>.txt`.

Chapter 24: Transaction Tracking Examples

This chapter contains a pseudocode example of how an application might be instrumented with ARM API calls, so that the transactions defined in the application can be monitored with Performance Collection Component or GlancePlus. This pseudocode example corresponds with the real time order processing scenario described in ["What is Transaction Tracking?" on page 336](#)

Several example transaction configuration files are included in this chapter, including one that corresponds with the real time order processing scenario.

Pseudocode for Real Time Order Processing

This pseudocode example includes the ARM API calls used to define transactions for the real time order processing scenario described in ["What is Transaction Tracking?" on page 336](#). This routine would be processed *each time* an operator answered the phone to handle a customer order. The lines containing the ARM API calls are highlighted with bold text in this example.

```
routine answer calls()
{
*****
* Register the transactions if first time in      *
*****

    if (transactions not registered)
    {
        appl_id = arm_init("Order Processing Application","*", 0,0,0)
        answer_phone_id = arm_getid(appl_id,"answer_phone","1st tran",0,0,0)
        if (answer_phone_id < 0)
            REGISTER OF ANSWER_PHONE FAILED - TAKE APPROPRIATE ACTION
        order_id = arm_getid(appl_id,"order","2nd tran",0,0,0)
        if (order_id < 0)
            REGISTER OF ORDER FAILED - TAKE APPROPRIATE ACTION
        check_id = arm_getid(appl_id,"check_db","3rd tran",0,0,0)
        if (check_id < 0)
            REGISTER OF CHECK DB FAILED - TAKE APPROPRIATE ACTION
    }
}
```

```
        update_id = arm_getid(appl_id,"update","4th tran",0,0,0)
        if (update_id < 0)
            REGISTER OF UPDATE FAILED - TAKE APPROPRIATE ACTION
    } if transactions not registered
*****
* Main transaction processing loop
*****
    while (answering calls)
    {
        if (answer_phone_handle = arm_start(answer_phone_id,0,0,0) < -1)
            TRANSACTION START FOR ANSWER_PHONE NOT REGISTERED
*****
* At this point the answer_phone transaction has *
* started. If the customer does not want to order, *
* end the call; otherwise, proceed with order. *
*****
        if (don't want to order)
            arm_stop(answer_phone_handle,ARM_FAILED,0,0,0)
            GOOD-BYE - call complete
        else
        {
*****
* They want to place an order - start an order now *
*****
            if (order_handle = arm_start(order_id,0,0,0) < -1)
                TRANSACTION START FOR ORDER FAILED
                take order information: name, address, item, etc.
*****
* Order is complete - end the order transaction *
*****
            if (arm_stop(order_handle,ARM_GOOD,0,0,0) < -1)
                TRANSACTION END FOR ORDER FAILED
*****
```

```
* order taken - query database for availability      *
*****
    if (query_handle = arm_start(query_id,0,0,0) < -1)
        TRANSACTION QUERY DB FOR ORDER NOT REGISTERED
    query the database for availability
*****
* database query complete - end query transaction  *
*****
    if (arm_stop(query_handle,ARM_GOOD,0,0,0) < -1)
        TRANSACTION END FOR QUERY DB FAILED
*****
* If the item is in stock, process order, and      *
* update inventory.                               *
*****
    if (item in stock)
        if (update_handle = arm_start(update_id,0,0,0) < -1)
            TRANSACTION START FOR UPDATE NOT REGISTERED
        update stock
*****
* update complete - end the update transaction    *
*****
    if (arm_stop(update_handle,ARM_GOOD,0,0,0) < -1)
        TRANSACTION END FOR ORDER FAILED
*****
* Order complete - end the call transaction       *
*****
    if (arm_stop(answer_phone_handle,ARM_GOOD,0,0,0) < -1)
        TRANSACTION END FOR ANSWER_PHONE FAILED
    } placing the order
    GOOD-BYE - call complete
    sleep("waiting for next phone call...zzz...")
    } while answering calls
    arm_end(appl_id, 0,0,0)
```

```
}      routine answer calls
```

Configuration File Examples

This section contains some examples of the transaction configuration file, `/var/opt/perf/ttd.conf`. For more information on the `ttd.conf` file and the configuration file keywords, see ["How Transaction Tracking Works" on page 341](#)

Example 1 (for Order Processing Pseudocode Example)

```
# The "*" entry below is used as the default if none of the  
# entries match a registered transaction name.
```

```
tran=* range=0.5,1,1.5,2,3,4,5,6,7 slo=1  
tran=answer_phone* range=0.5,1,1.5,2,3,4,5,6,7 slo=5  
tran=order* range=0.5,1,1.5,2,3,4,5,6,7 slo=5  
tran=query_db* range=0.5,1,1.5,2,3,4,5,6,7 slo=5
```

Example 2

```
# The "*" entry below is used as the default if none of the  
# entries match a registered transaction name.
```

```
tran=* range=1,2,3,4,5,6,7,8 slo=5
```

```
# The entry below is for the only transaction being  
# tracked in this application. The "*" has been inserted  
# at the end of the tran name to catch any possible numbered  
# transactions. For example "First_Transaction1",  
# "First_Transaction2", etc.
```

```
tran=First_Transaction* range=1,2.2,3.3,4.0,5.5,10 slo=5.5
```

Example 3

```
# The "*" entry below is used as the default if none of the  
# entries match a registered transaction name.
```

```
tran=*  
tran=Transaction_One range=1,10,20,30,40,50,60 slo=30
```

Example 4

```
tran=FactoryStor* range=0.05, 0.10, 0.15 slo=3  
# The entries below shows the use of an application name.  
# Transactions are grouped under the application name. This  
# example also shows the use of less than 10 ranges and  
# optional use of "slo."
```

```
[Inventory]  
tran=In_Stock range=0.001, 0.004, 0.008  
tran=Out_Stock range=0.001, 0.005  
tran>Returns range=0.1, 0.3, 0.7
```

```
[Pers]  
tran=Acctg range=0.5, 0.10, slo=5  
tran=Time_Cards range=0.010, 0.020
```

Chapter 25: Advanced Features

This chapter describes how Performance Collection Component uses the following ARM 2.0 API features:

- data types
- user-defined metrics
- scope instrumentation

How Data Types Are Used

The table below describes how data types are used in Performance Collection Component. It is a supplement to “Data Type Definitions” in the “Advanced Topics” section of the [Application Response Measurement 2.0 API Guide](#).

Table 3: Data type usage in Performance Collection Component

ARM_Counter32	Data is logged as a 32-bit integer.
ARM_Counter64	Data is logged as a 32-bit integer with type casting.
ARM_CntrDivr32	Makes the calculation and logs the result as a 32-bit integer.
ARM_Gauge32	Data is logged as a 32-bit integer.
ARM_Gauge64	Data is logged as a 32-bit integer with type casting.
ARM_GaugeDivr32	Makes the calculation and logs the result as a 32-bit integer.
ARM_NumericID32	Data is logged as a 32-bit integer.
ARM_NumericID64	Data is logged as a 32 bit integer with type casting.
ARM_String8	Ignored.
ARM_String32	Ignored.

Performance Collection Component does not log string data. Because Performance Collection Component logs data every five minutes, and what is logged is the summary of the activity for that interval, it cannot summarize the strings provided by the application.

Performance Collection Component logs the Minimum, Maximum, and Average for the first six usable user-defined metrics. If your ARM-instrumented application passes a Counter32, a String8, a NumericID 32, a Gauge32, a Gauge64, a Counter64, a NumericID64, a String32, and a GaugeDivr32, Performance Collection Component logs the Min, Max, and Average over the five-minute interval for the Counter32, NumericID32, Gauge32, Gauge64, NumericID32 and NumericID64 as 32-bit integers. The String8 and String32 are ignored because strings cannot be summarized in Performance Collection

Component. The GaugeDivr32 is also ignored because only the first six usable user-defined metrics are logged. (For more examples, see the next section, [User-Defined Metrics](#))

User-Defined Metrics

This section is a supplement to “Application-Defined Metrics” under “Advanced Topics” in the [Application Response Measurement 2.0 API Guide](#). It contains some examples about how Performance Collection Component handles user-defined metrics (referred to as application-defined metrics in ARM). The examples in Table 4 show what is logged if your program passes the following data types.

Table 4: Examples of What is Logged with Specific Program Data Types

...what your program passes in	...what is logged
EXAMPLE 1 String8 Counter32 Gauge32 CntrDivr32	Counter32 Gauge32 CntrDivr32
EXAMPLE 2 String32 NumericID32 NumericID64	NumericID32 NumericID64
EXAMPLE 3 NumericID32 String8 NumericID64 Gauge32 String32 Gauge64	NumericID32 NumericID64 Gauge32 Gauge64
EXAMPLE 4 String8 String32	(nothing)
EXAMPLE 5 Counter32 Counter64	Counter32 Counter64

Table 4: Examples of What is Logged with Specific Program Data Types, continued

...what your program passes in	...what is logged
CntrDivr32	CntrDivr32
Gauge32	Gauge32
Gauge64	Gauge64
NumericID32	NumericID32
NumericID64	

Because Performance Collection Component cannot summarize strings, no strings are logged.

In example 1, only the counter, gauge, and counter divisor are logged.

In example 2, only the numerics are logged.

In example 3, only the numerics and gauges are logged.

In example 4, nothing is logged.

In example 5, because only the first six user-defined metrics are logged, NumericID64 is not logged.

scope Instrumentation

The `scope` data collector has been instrumented with ARM API calls. When Performance Collection Component starts, `scope` automatically starts logging two transactions, `Scope_Get_Process_Metrics` and `Scope_Get_Global_Metrics`. Both transactions will be in the HP Performance Tools application.

Transaction data is logged every five minutes so you will find that five `Get Process` transactions (one transaction per minute) have completed during each interval. The `Scope_Get_Process_Metrics` transaction is instrumented around the processing of process data. If there are 200 processes on your system, the `Scope_Get_Process_Metrics` transaction should take longer than if there are only 30 processes on your system.

The `Scope_Get_Global_Metrics` transaction is instrumented around the gathering of all five-minute data, including global data. This includes `global`, `application`, `disk`, `transaction`, and other data types.

To stop the logging of process and global transactions data, remove or comment out the entries for the `scope` transactions in the `ttd.conf` file.

Chapter 26: Transaction Libraries

This appendix discusses:

- the Application Response Measurement library (`libarm`)
- C compiler option examples by platform
- the Application Response Measurement NOP library (`libarmNOP`)
- using Java wrappers

ARM Library (`libarm`)

With Performance Collection Component and GlancePlus, the environment is set up to make it easy to compile and use the ARM facility. The libraries needed for development are located in `/opt/perf/lib/`. See the next section in this appendix for specific information about compiling.

The library files listed in Table 5 exist on an HP-UX 11.11 and beyond Performance Collection Component and GlancePlus installation:

Table 5: HP-UX 11.11 and Beyond Performance Collection Component and GlancePlus Library Files

<code>/opt/perf/lib/</code>	<code>libarm.0</code>	HP-UX 10.X compatible shared library for ARM (not thread safe). If you execute a program on HP-UX 11 that was linked on 10.20 with <code>-larm</code> , the 11.0 loader will automatically reference this library.
	<code>libarm.1</code>	HP-UX 11 compatible shared library (thread safe). This will be referenced by programs that were linked with <code>-larm</code> on HP-UX releases. If a program linked on 10.20 references this library, (for example, if it was not linked with <code>-L /opt/perf/lib</code> , it may abort with an error such as <code>"/usr/lib/dld.sl: Unresolved symbol: _thread_once (code) from libtt.sl"</code> .
	<code>libarm.sl</code>	A symbolic link to <code>libarm.1</code>
	<code>libarmNOP.sl</code>	"No-operation" shared library for ARM (the API calls succeed but do nothing; used for testing and on systems that do not have Performance Collection Component installed.

/opt/perf/examples/arm	libarmjava.sl	32-bit shared library for ARM.
/opt/perf/examples/arm/arm64	libarmjava.sl	64-bit shared library for ARM.
/opt/perf/lib/pa20_64/	Note that these files will be referenced automatically by programs compiled on HP-UX 11 with the +DD64 compiler option.	
	libarm.sl	64-bit shared library for ARM.
	libarmNOP.sl	64-bit “no-operation” shared library for ARM (the API calls succeed but do nothing; used for testing and on systems that do not have Performance Collection Component installed).

The additional library files listed in Table 6 exist on an IA64 HP-UX installation:

Table 6: HP-UX IA64 Library Files

/opt/perf/lib/hpux32/	libarm.so.1	IA64/32-bit shared library for ARM.
/opt/perf/lib/hpux64/	libarm.so.1	IA64/64-bit shared library for ARM.
/opt/perf/examples/arm	libarmjava.so	32-bit shared library for ARM.
/opt/perf/examples/arm/arm64	libarmjava.so	64-bit shared library for ARM.

Because the ARM library makes calls to HP-UX that may change from one version of the operating system to the next, programs should link with the shared library version, using `-larm`. Compiling an application that has been instrumented with ARM API calls and linking with the archived version of the ARM library (`-Wl, -a archive`) is not supported. (For additional information, see ["Transaction Tracking Daemon \(ttdd\)" on page 343](#) in Chapter 2.

The library files that exist on an AIX operating system with Performance Collection Component and GlancePlus installation are as follows.

Table 7: AIX Library Files

/usr/lpp/perf/lib/	libarm.a	32-bit shared ARM library (thread safe). This library is referenced by programs linked with <code>-larm</code> .
/usr/lpp/perf/lib	libarmNOP.a	32-bit shared library for ARM. This library is used for testing on systems that do not have Performance Agent/Performance Collection Component installed.
/usr/lpp/perf/lib64/	libarm.a	64-bit shared ARM library (thread safe). This library is referenced by programs linked with <code>-larm</code> .
/usr/lpp/perf/lib64	libarmNOP.a	64-bit shared library for ARM. This library is used for testing on systems that do not

		have Performance Agent/Performance Collection Component installed.
/usr/lpp/perf/examples/arm	libarmjava.a	32-bit shared library for ARM
/usr/lpp/perf/examples/arm/arm64	libarmjava.a	64-bit shared library for ARM.
/usr/lpp/perf/lib/	libarmns.a	32-bit archived ARM library. Functionality wise this is same as 32 bit libarm.a.
/usr/lpp/perf/lib64/	libarmns.a	64-bit archived ARM library. Functionality wise this is same as 64 bit libarm.a.

The library files that exist on a Solaris operating system with Performance Collection Component and GlancePlus installation are as follows.

Table 8: Solaris Library Files for 32-bit programs

/opt/perf/lib/	libarm.so	32-bit shared ARM library (thread safe). This library is referenced by programs linked with -larm.
	libarmNOP.so	32-bit shared library for ARM. This library is used for testing on systems that do not have Performance Collection Component installed.

Table 9: Solaris Library Files for Sparc 64-bit programs

/opt/perf/lib/sparc_64/	libarm.so	64-bit shared ARM library (thread safe). This library is referenced by programs linked with -larm.
	libarmNOP.so	64-bit shared library for ARM This library is used for testing on systems that do not have Performance agent/Performance Collection Component installed.
/opt/perf/examples/arm	libarmjava.so	32-bit shared library for ARM.
/opt/perf/examples/arm/arm64	libarmjava.so	64-bit shared library for ARM.

Table 10: Solaris Library Files for x86 64-bit programs

/opt/perf/lib/x86_64/	libarm.so	64-bit shared ARM library (thread safe). This library is referenced by programs linked with -larm.
	libarmNOP.so	64-bit shared library for ARM This library is used for testing on systems that do not have Performance agent installed.
/opt/perf/examples/arm/arm64	libarmjava.so	32-bit shared library for ARM.
/opt/perf/examples/arm/arm64	libarmjava.so	64-bit shared library for ARM.

Note: You must compile 64-bit programs using `-xarch=generic64` command-line parameter along with the other parameters provided for 32-bit programs.

The library files that exist on a Linux operating system with Performance Collection Component and GlancePlus installation are as follows.

Table 11: Linux Library Files

/opt/perf/lib/	libarm.so	32-bit shared ARM library (thread safe). This library is referenced by programs linked with <code>-larm</code> .
	libarmNOP.so	32-bit shared library for ARM. This library is used for testing on systems that do not have Performance Collection Component installed.
/opt/perf/lib/	libarm.so	64-bit shared ARM library (thread safe). This library is referenced by programs linked with <code>-larm</code> .
	libarmNOP.so	64-bit shared library for ARM. This library is used for testing on systems that do not have Performance Collection Component installed.
/opt/perf/examples/arm	libarmjava.so	32-bit shared library for ARM.
/opt/perf/examples/arm/arm64	libarmjava.so	64-bit shared library for ARM.

Note: For Linux 2.6 IA 64 bit 32 bit `libarm.so` and `libarmjava.so` are not implemented.

C Compiler Option Examples by Platform

The `arm.h` include file is located in `/opt/perf/include/`. For convenience, this file is accessible via a symbolic link from `/usr/include/` as well. This means that you do not need to use `“-I/opt/perf/include/”` (although you may). Likewise, `libarm` resides in `/opt/perf/lib/` but is linked from `/usr/lib/`. You should always use `“-L/opt/perf/lib/”` when building ARMed applications.

- For Linux:
 The following example shows a compile command for a C program using the ARM API.

```
cc myfile.c -o myfile -I /opt/perf/include -L -Xlinker -rpath -Xlinker /opt/perf/lib
```

- For 64-bit programs on Linux:

```
cc -m64 myfile.c -o myfile -I /opt/perf/include -L -Xlinker -rpath -Xlinker /opt/perf/lib64
```

- For HP-UX:
For HP-UX releases 11.2x on IA64 platforms, change the `-L` parameter from `-L/opt/perf/lib` to `-L/opt/perf/lib/hpux32` for 32-bit IA ARMed program compiles, and to `-L/opt/perf/lib/hpux64` for 64-bit IA program compiles using ARM.

The following example shows a compile command for a C program using the ARM API.

```
cc myfile.c -o myfile -I /opt/perf/include -L /opt/perf/lib -larm
```

- For Sun Solaris:
The following example works for Performance Collection Component and GlancePlus on Sun Solaris:

```
cc myfile.c -o myfile -I /opt/perf/include -L /opt/perf/lib -larm -lnsl
```

- For 64-bit Sparc programs on Sun Solaris:
The following example works for Performance Collection Component and 64-bit programs on Sun Solaris:

```
cc -xarch=generic64 myfile.c -o myfile -I /opt/perf/include -L /opt/perf/lib/sparc_64 -larm -lnsl
```

- For 64-bit x86 programs on Sun Solaris:
The following example works for Performance agent and 64-bit programs on Sun Solaris:

```
cc -xarch=generic64 myfile.c -o myfile -I /opt/perf/include -L /opt/perf/lib/x86_64 -larm -lnsl
```

- For IBM AIX:
The file placement on IBM AIX differs from the other platforms (`/usr/lpp/perf/` is used instead of `/opt/perf/`), therefore the example for IBM AIX is different from the examples of other platforms:

```
cc myfile.c -o myfile -I /usr/lpp/perf/include -L /usr/lpp/perf/lib -larm
```

- For 64-bit programs on IBM AIX:
The following example works for Performance agent and 64-bit programs on IBM AIX:

```
cc -q64 myfile.c -o myfile -I /usr/lpp/perf/include -L /usr/lpp/perf/lib64 -larm
```

Note: For C++ compilers, the `-D_PROTOTYPES` flag may need to be added to the compile command in order to reference the proper declarations in the `arm.h` file.

ARM NOP Library

The “no-operation” library (named `libarmNOP.*` where `*` is `sl`, `so`, or `a`, depending on the OS platform) is shipped with Performance Collection Component and Glance. This shared library does nothing except return valid status for every ARM API call. This allows an application that has been instrumented with ARM to run on a system where Performance Collection Component or GlancePlus is not installed.

To run your ARM-instrumented application on a system where Performance Collection Component or GlancePlus is not installed, copy the NOP library and name it `libarm.sl` (`libarm.so`, or `libarm.a` depending on the platform) in the appropriate directory (typically, `/<InstallDir>/lib/`). When Performance Collection Component or GlancePlus is installed, it will overwrite this NOP library with the correct functional library (which is not removed as the other files are). This ensures that

instrumented programs will not abort when Performance Collection Component or GlancePlus is removed from the system.

Using the Java Wrappers

The Java Native Interface (JNI) wrappers are functions created for your convenience to allow the Java applications to call the HP ARM2.0 API. These wrappers (`armapi.jar`) are included with the ARM sample programs located in the `/<InstallDir>/examples/arm/` directory. `InstallDir` is the directory in which Performance Collection Component is installed.

Examples

Examples of the Java wrappers are located in the `/<InstallDir>/examples/arm/` directory. This location also contains a README file, which explains the function of each wrapper.

Setting Up an Application (`arm_init`)

To set up a new application, make a new instance of `ARMApplication` and pass the name and the description for this API. Each application needs to be identified by a unique name. The `ARMApplication` class uses the C – function `arm_init`.

Syntax:

```
ARMApplication myApplication =new ARMApplication("name","description")
```

Setting Up a Transaction (`arm_getid`)

To set up a new transaction, you can choose whether or not you want to use user-defined metrics (UDMs). The Java wrappers use the C – function `arm_getid`.

Setting Up a Transaction With UDMs

If you want to use UDMs, you must first define a new `ARMTranDescription`. `ARMTranDescription` builds the Data Buffer for `arm_getid`. (See also the `jprimeudm.java` example.)

Syntax:

```
ARMTranDescription myDescription =  
    new ARMTranDescription("transactionName","details");
```

If you don't want to use details, you can use another constructor:

Syntax:

```
ARMTranDescription myDescription =
```

```
new ARMTranDescription("transactionName");
```

Adding the Metrics

Metric 1-6:

Syntax:

```
myDescription.addMetric(metricPosition, metricType, metricDescription);
```

Parameters:

metricPosition: 1-6

metricType: ARMConstants.ARM_Counter32

ARMConstants.ARM_Counter64 ARMConstants.ARM_CntrDivr32

ARMConstants.ARM_Gauge32 ARMConstants.ARM_Gauge64

ARMConstants.ARM_GaugeDivr32 ARMConstants.ARM_NumericID32

ARMConstants.ARM_NumericID64 ARMConstants.ARM_String8

Metric 7:

Syntax:

```
myDescription.addStringMetric("description");
```

Then you can create the Transaction:

Syntax:

```
myApplication.createTransaction(myDescription);
```

Setting the Metric Data

Metric 1-6:

Syntax:

```
myTransaction.setMetricData(metricPosition, metric);
```

Examples for "Metric"

```
ARMGauge32Metric metric = new ARMGauge32Metric(start);
```

```
ARMCounter32Metric metric = new ARMCounter32Metric(start);
```

```
ARMCntrDivr32Metric metric = new ARMCntrDivr32Metric(start, 1000);
```

Metric 7:

Syntax:

```
myTransaction.setStringMetricData(text);
```

Setting Up a Transaction Without UDMs

When you set up a transaction without UDMs, you can immediately create the new transaction. You can choose whether or not to specify details.

With Details

Syntax:

```
ARMTransaction myTransaction =  
myApplication.createTransaction("Transactionname", "details");
```

Without Details

Syntax:

```
ARMTransaction myTransaction =  
myApplication.createTransaction("Transactionname");
```

Setting Up a Transaction Instance

To set up a new transaction instance, make a new instance of `ARMTransactionInstance` with the method `createTransactionInstance()` of `ARMTransaction`.

Syntax:

```
ARMTransactionInstance myTranInstance =  
myTransaction.createTransactionInstance();
```

Starting a Transaction Instance (arm_start)

To start a transaction instance, you can choose whether or not to use correlators. The following methods call the C – function `arm_start` with the relevant parameters.

Starting the Transaction Instance Using Correlators

When you use correlators, you must distinguish between getting and delivering a correlator.

Requesting a Correlator

If your transaction instance wants to request a correlator, the call is as follows (see also the `jcorrelators.java` example).

Syntax:


```
int status = myTranInstance.startTranWithCorrelator();
```

Passing the Parent Correlator

If you already have a correlator from a previous transaction and you want to deliver it to your transaction, the syntax is as follows:

Syntax

```
int status = startTran(parent);
```

Parameter

parent is the delivered correlator. In the previous transaction, you can get the transaction instance correlator with the method `getCorrelator()`.

Requesting and Passing the Parent Correlator

If you already have a correlator from a previous transaction and you want to deliver it to your transaction and request a correlator, the syntax is as follows:

Syntax:

```
int status = myTranInstance.startTranWithCorrelator(parent);
```

Parameter:

parent is the delivered correlator. In the previous transaction, you can get the transaction instance correlator with the method `getCorrelator()`.

Retrieving the Correlator Information

You can retrieve the transaction instance correlator using the `getCorrelator()` method as follows:

Syntax:

```
ARMTranCorrelator parent = myTranInstance.getCorrelator();
```

Starting the Transaction Instance Without Using Correlators

When you do not use correlators, you can start your transaction instance as follows:

Syntax:

```
int status = myTranInstance.startTran();
```

startTran returns a unique handle to status, which is used for the update and stop.

Updating Transaction Instance Data

You can update the UDMs of your transaction instance any number of times between the start and stop. This part of the wrappers calls the C – function `arm_update` with the relevant parameters.

Updating Transaction Instance Data With UDMs

When you update the data of your transaction instance with UDMs, first, you must set the new data for the metric. For example,

```
metric.setData(value) for ARM_Counter32 ARM_Counter64, ARM_Gauge32, ARM_Gauge64, ARM_NumericID32, ARM_NumericID64
```

```
metric.setData(value,value) for ARM_CntrDivr32 and , ARM_GaugeDivr32
```

```
metric.setData(string) for ARM_String8 and ARM_String32
```

Then you can set the metric data to new (like the examples in the ["Setting the Metric Data" on page 375](#) section) and call the update:

Syntax:

```
myTranInstance.updateTranInstance();
```

Updating Transaction Instance Data Without UDMs

When you update the data of your transaction instance without UDMs, you just call the update. This sends a “heartbeat” indicating that the transaction instance is still running.

Syntax:

```
myTranInstance.updateTranInstance();
```

Providing a Larger Opaque Application Private Buffer

If you want to use the second buffer format, you must pass the byte array to the update method. (See the [Application Response Measurement 2.0 API Guide](#).)

Syntax:

```
myTranInstance.updateTranInstance(byteArray);
```

Stopping the Transaction Instance (arm_stop)

To stop the transaction instance, you can choose whether or not to stop it with or without a metric update.

Stopping the Transaction Instance With a Metric Update

To stop the transaction instance with a metric update, call the method `stopTranInstanceWithMetricUpdate`.

Syntax:

```
myTranInstance.stopTranInstanceWithMetricUpdate  
transactionCompletionCode);
```

Parameter:

The transaction Completion Code can be:

<code>ARMConstants.ARM_GOOD.</code>	Use this value when the operation ran normally and as expected.	<code>ARMConstants.ARM_GOOD.</code>
<code>ARMConstants.ARM_ABORT.</code>	Use this value when there is a fundamental failure in the system.	<code>ARMConstants.ARM_ABORT.</code>
<code>ARMConstants.ARM_FAILED.</code>	Use this value in applications where the transaction worked properly, but no result was generated.	<code>ARMConstants.ARM_FAILED.</code>

These methods use the C – function `arm_stop` with the requested parameters.

Stopping the Transaction Instance Without a Metric Update

To stop the transaction instance without a metric update, you can use the method `stopTranInstance`.

Syntax:

```
myTranInstance.stopTranInstance(transactionCompletionCode);
```

Using Complete Transaction

The Java wrappers can use the `arm_complete_transaction` call. This call can be used to mark the end of a transaction that has lasted for a specified number of nanoseconds. This enables the real time

integration of transaction response times measured outside of the ARM agent.

In addition to signaling the end of a transaction instance, additional information about the transaction (UDMs) can be provided in the optional data buffer.

(See also the `jcomplete.java` example.)

Using Complete Transaction With UDMs:

Syntax:

```
myTranInstance.completeTranWithUserData(status,responseTime;
```

Parameters:

status	<ul style="list-style-type: none">• <code>ARMConstants.ARM_GOOD</code> Use this value when the operation ran normally and as expected.• <code>ARMConstants.ARM_ABORT</code> Use this value when there was a fundamental failure in the system.• <code>ARMConstants.ARM_FAILED</code> Use this value in applications where the transaction worked properly, but no result was generated.
responseTime	This is the response time of the transaction in nanoseconds.

Using Complete Transaction Without UDMs:

Syntax:

```
myTranInstance.completeTran(status,responseTime);
```

Further Documentation

For further information about the Java classes, see the doc folder in the `/<InstallDir>/examples/arm/` directory, which includes html-documentation for every Java class. Start with `index.htm`.

Chapter 27: Logging and Tracing

You can diagnose and troubleshoot problems in the HP Operations agent by using the logging and tracing mechanisms. The HP Operations agent stores error, warning, and general messages in log files for easy analysis.

The tracing mechanism helps you trace specific problems in the agent's operation; you can transfer the trace files generated by the tracing mechanism to HP Support for further analysis.

Logging

The HP Operations agent writes warning and error messages and informational notifications in the `System.txt` file on the node. The contents of the `System.txt` file reveal if the agent is functioning as expected. You can find the `System.txt` file in the following location:

On Windows

`%ovdatadir%\log`

On UNIX/Linux

`/var/opt/OV/log`

In addition, the HP Operations agent adds the status details of the Performance Collection Component and coda in the following files:

On Windows

- `%ovdatadir%\status.scope`
- `%ovdatadir%\status.perfalarm`
- `%ovdatadir%\status.ttd`
- `%ovdatadir%\status.mi`
- `%ovdatadir%\status.perfd-<port>`

Tip: In this instance, *<port>* is the port used by `perfd`. By default, `perfd` uses the port 5227. To change the default port of `perfd`, see [Configuring the RTMA Component](#).

`%ovdatadir%\log\coda.txt`

On UNIX/Linux

- `/var/opt/perf/status.scope`
- `/var/opt/perf/status.perfalarm`
- `/var/opt/perf/status.ttd`
- `/var/opt/perf/status.mi`
- `/var/opt/perf/status.perfd`

- *Only on vMA.* /var/opt/perf/status.viserver
- /var/opt/OV/log/coda.txt

Configure the Logging Policy

The `System.txt` file can grow up to 1 MB in size, and then the agent starts logging messages in a new version of the `System.txt` file. You can configure the message logging policy of the HP Operations agent to restrict the size of the `System.txt` file.

To modify the default logging policy, follow these steps:

1. Log on to the node.
2. Go to the following location:
On Windows
`%ovdatadir%conf\xpl\log`
On UNIX/Linux
`/var/opt/OV/conf/xpl/log`
3. Open the `log.cfg` file with a text editor.
4. The `BinSizeLimit` and `TextSizeLimit` parameters control the byte size and number of characters of the `System.txt` file. By default, both the parameters are set to 1000000 (1 MB and 1000000 characters). Change the default values to the desired values.
5. Save the file.
6. Restart the Operations Monitoring Component with the following commands:
 - a. `ovc -kill`
 - b. `ovc -start`

Tracing

Before you start tracing an HP Operations agent application, you must perform a set of prerequisite tasks, which includes identifying the correct application to be traced, setting the type of tracing, and generating a trace configuration file (if necessary).

Before you begin tracing an HP Operations agent process, perform the following tasks:

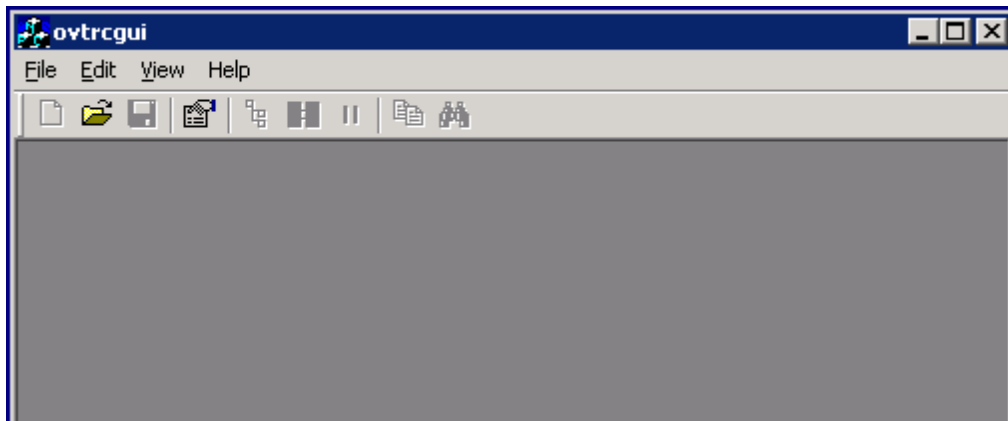
1. [Identify the Application](#)
2. [Set the Tracing Type](#)
3. *Optional.* [Create the Configuration File](#)

Identify the Application

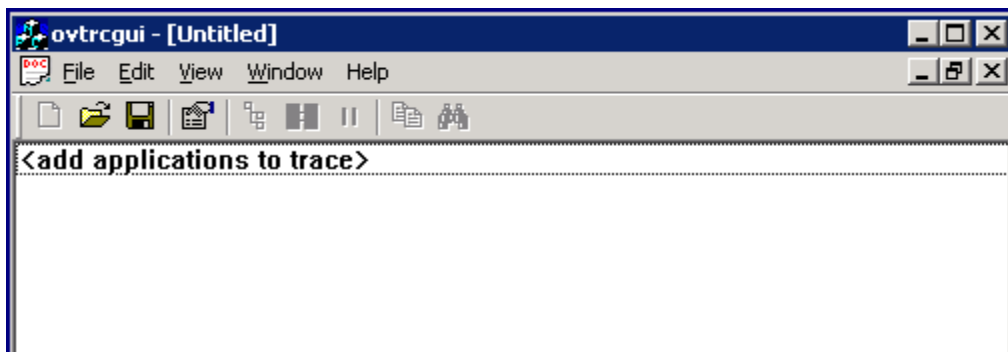
On the managed system, identify the HP Software applications that you want to trace. Use the `ovtrccfg -vc` option to view the names of all trace-enabled applications and the components and categories defined for each trace-enabled application.

Alternatively, you can use the ovtrcgui utility to view the list of trace-enabled applications. To use the ovtrcgui utility to view the list of trace-enabled applications, follow these steps:

1. Run the ovtrcgui.exe file from the %OvInstallDir%\support directory. The ovtrcgui window opens.

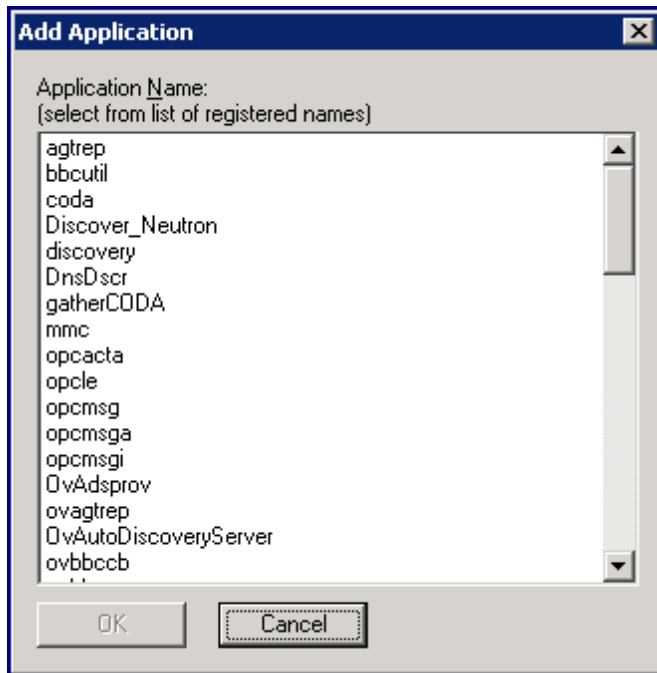


2. In the ovtrcgui window, click **File** → **New** → **Trace Configuration**. A new trace configuration editor opens.



3. In the ovtrcgui window, click **Edit** → **Add Application**. Alternatively, right-click on the editor, and

then click **Add Application**. The Add Application window opens.



The Add Application window presents a list of available trace-enabled applications.

Set the Tracing Type

Before you enable the tracing mechanism, decide and set the type of tracing that you want to configure with an application. To set the type of tracing, follow these steps:

Determine the type of tracing (static or dynamic) you want to configure, and then follow these steps:

1. Go to the location `<data_dir>/conf/xpl/trc/`
2. Locate the `<application_name>.ini` file. If the file is present, go to step 3 below. If the `<application_name>.ini` file is not present, follow these steps:
 - Create a new file with a text editor.
 - Add the following properties to the file in the given order: DoTrace, UpdateTemplate, and DynamicTracing.

Tip: Do not list the properties in a single line. List every property in a new line. For example:

```
DoTrace=  
UpDateTemplate=  
DynamicTracing=
```

- Save the file.
3. Open the `<application_name>.ini` file with a text editor.

4. To enable the static tracing, make sure that the DoTrace property is set to ON and the DynamicTracing property is set to OFF.
5. To enable the dynamic tracing, make sure that the DoTrace and DynamicTracing properties are set to ON.
6. Make sure that the UpdateTemplate property is set to ON.
7. Save the file.

For the dynamic trace configuration, you can enable the tracing mechanism even after the application starts. For the static trace configuration, you must enable the tracing mechanism before the application starts.

Introduction to the Trace Configuration File

Syntax

TCF Version `<version_number>`

APP: "`<application_name>`"

SINK: File "`<file_name>`" "maxfiles=[`1..100`];maxsize=[`0..1000`];"

TRACE: "`<component_name>`" "`<category_name>`" `<keyword_list>`

Each line of the syntax is described in detail in the following sections.

Create the Configuration File

If you want to enable the tracing mechanism without the help of a configuration file, skip this section and go to the section *Enabling Tracing and Viewing Trace Messages with the Command-Line Tools* below.

You can create the trace configuration file with the command-line tool `ovtrccfg`, with a text editor, or with the `ovtrcgui` utility (only on Windows nodes).

Enabling Tracing and Viewing Trace Messages with the Command-Line Tools

The procedure outlined below covers the general sequence of steps required to enable tracing. To enable the tracing mechanism, follow these steps:

1. Make a trace configuration request using `ovtrccfg`.
`ovtrccfg -cf <configuration_file_name>`
where `<configuration_file_name>` is the name of the trace configuration file created in the section above *Create the Configuration File*.

Note: If you do not want to use a trace configuration file, you can enable tracing with the following command:

```
ovtrccfg -app <application>[-cm <component>]
```

2. If you configure the static tracing mechanism, start the application that you want to trace.
3. Run the application specific commands necessary to duplicate the problem that you want to trace. When the desired behavior has been duplicated, tracing can be stopped.
4. Make a trace monitor request using `ovtrcmon`.
To monitor trace messages, run one of the following commands or a similar command using additional `ovtrcmon` command options:
 - To monitor trace messages from `/opt/OV/bin/trace1.trc` and direct trace messages to a file in the text format:
`ovtrcmon -fromfile /opt/OV/bin/trace1.trc -tofile /tmp/traceout.txt`
 - To view trace messages from `/opt/OV/bin/trace1.trc` in the verbose format:
`ovtrcmon -fromfile /opt/OV/bin/trace1.trc -verbose`
 - To view trace messages from `/opt/OV/bin/trace1.trc` in the verbose format and direct the trace message to a file:
`ovtrcmon -fromfile /opt/OV/bin/trace1.trc -short > /tmp/traces.trc`
5. To stop or disable tracing using `ovtrccfg`, run the following command:
`ovtrccfg -off`
6. Collect the trace configuration file and the trace output files. Evaluate the trace messages or package the files for transfer to HP Software Support Online for evaluation. There may be multiple versions of the trace output files on the system. The `Maxfiles` option allows the tracing mechanism to generate multiple trace output files. These files have the extension `.trc` and the suffix `n` (where `n` is an integer between 1 and 99999).

Enabling Tracing and Viewing Trace Messages with the Tracing GUI

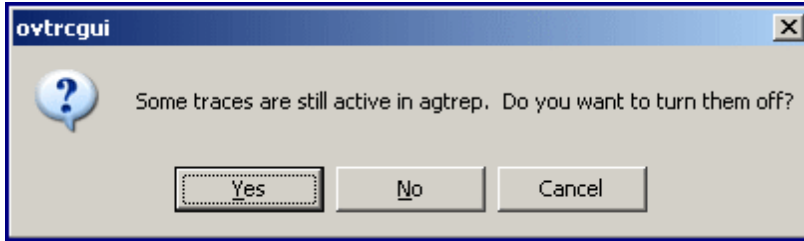
On the Windows nodes, you can use the `ovtrcgui` utility to configure tracing and view the trace messages.

Enable the Tracing Mechanism

To enable the tracing mechanism with the `ovtrcgui` utility and without the help of a trace configuration file, follow these steps:

1. Follow Step 1 through Step 6 in [Using the Tracing GUI](#).
2. Close the trace configuration editor.
3. Click **No** when prompted to save changes to Untitled.

The following message appears:



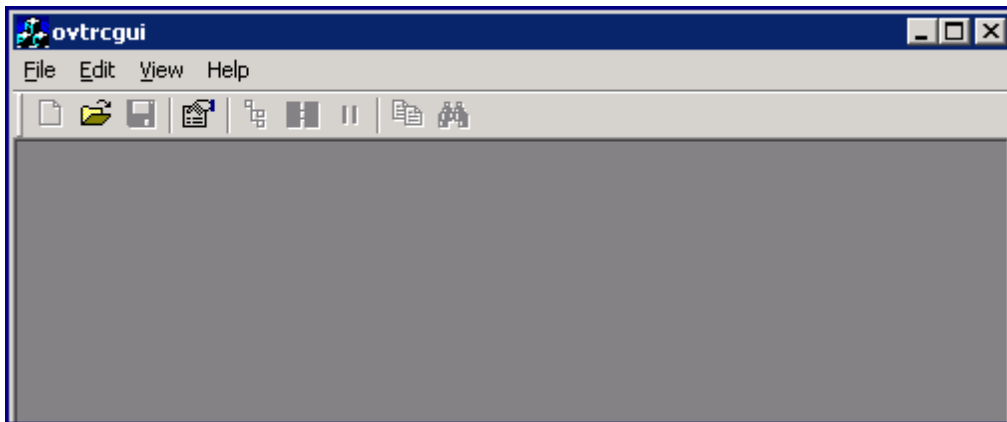
4. Click **No**. If you click **Yes**, the ovtrcgui utility immediately disables the tracing mechanism.

To enable the tracing mechanism with the ovtrcgui utility using a trace configuration file, go to the location on the local system where the trace configuration file is available, and then double-click on the trace configuration file. Alternatively, open the ovtrcgui utility, click **File** → **Open**, select the trace configuration file, and then click **Open**.

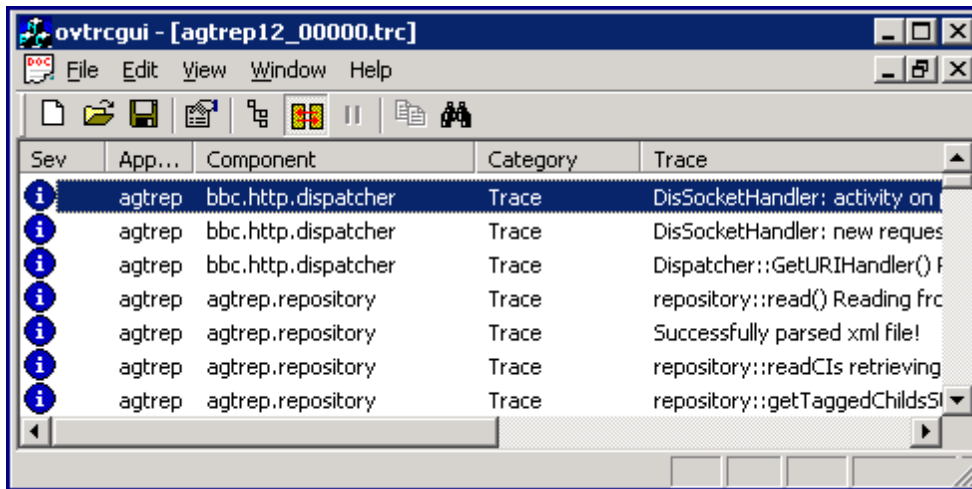
View Trace Messages

To view the trace output files with the ovtrcgui utility, follow these steps:

1. Run the ovtrcgui.exe file from the %OvInstallDir%\support directory. The ovtrcgui window opens.

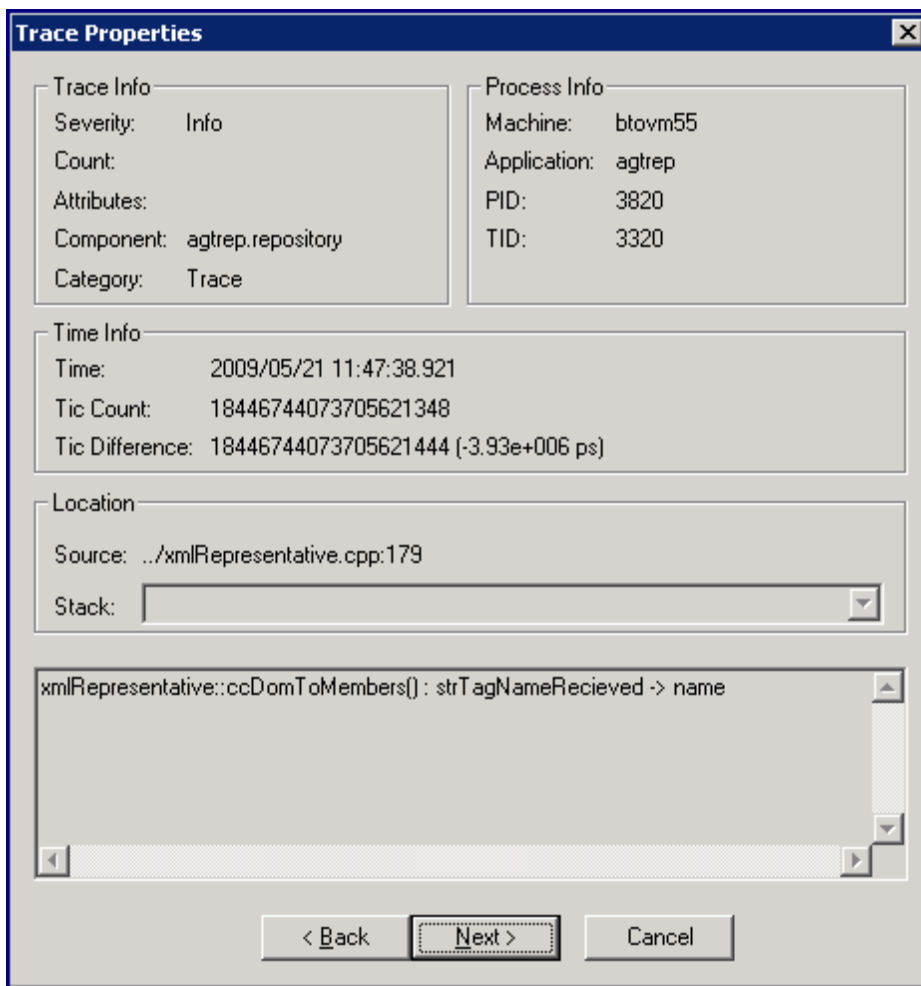


2. Click **File** → **Open**. The Open dialog box opens.
3. Navigate to the location where the trace output file is placed, select the .trc file, and then click **Open**. The ovtrcgui utility shows the contents of the .trc file.



Every new line in the .trc file represents a new trace message.

4. Double-click a trace message to view the details. The Trace Properties window opens.



The Trace Properties window presents the following details:




- Trace Info:
 - *Severity*: The severity of the trace message.
 - *Count*: The serial number for the message.
 - *Attributes*: The attribute of the trace message.
 - *Component*: Name of the component that issues the trace message.
 - *Category*: An arbitrary name assigned by the traced application.
 - Process Info:
 - *Machine*: Hostname of the node.
 - *Application*: Name of the traced application.
 - *PID*: Process ID of the traced application.
 - *TID*: Thread ID of the traced application.
 - Time Info:
 - *Time*: The local-equivalent time and date of the trace message.
 - *Tic count*: A high-resolution elapsed time.
 - *Tic difference*:
 - Location
 - *Source*: Line number and file name of the source generating the trace.
 - *Stack*: A description of the calling stack in the traced application.
5. Click **Next** to view the next trace message.
 6. After viewing all the trace messages, click **Cancel**.

Use the Trace List View

By default, the `ovtrcgui` utility displays the trace messages for a trace file in the trace list view. The trace list view presents the trace messages in a tabular format.

The trace list view presents every trace message with the help of the following columns:

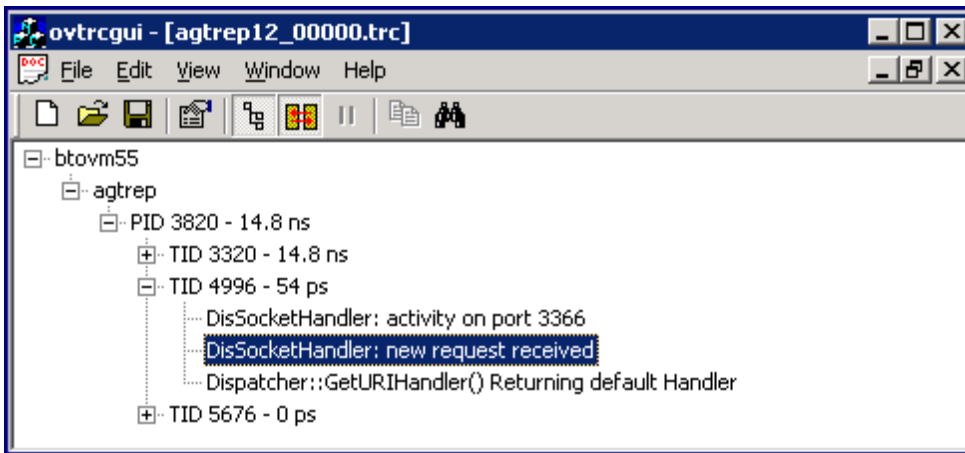
Table 13: Trace List View


Column	Description
Severity	Indicates the severity of the trace message. The view uses the following icons to display the severity of the messages:
	Info 
	Warning 
	Error 
Application	Displays the name of the traced application.
Component	Displays the name of the component of the traced application that generated the trace

Column	Description
	message.
Category	Displays the category of the trace message.
Trace	Displays the trace message text.

Use the Procedure Tree View

You can view the trace messages in a structured format in the procedure tree view. The procedure tree view sorts the messages based on the process IDs and thread IDs and presents the data in the form of a tree view.

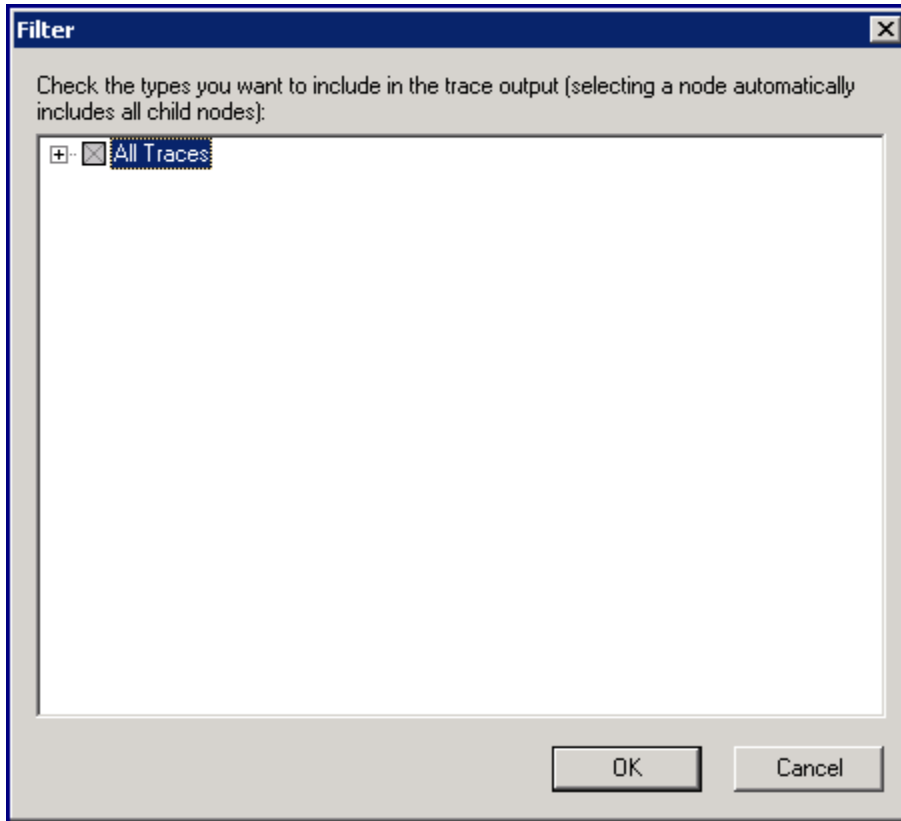


You can expand the process IDs and thread IDs to view trace messages. To go back to the trace list view, click .

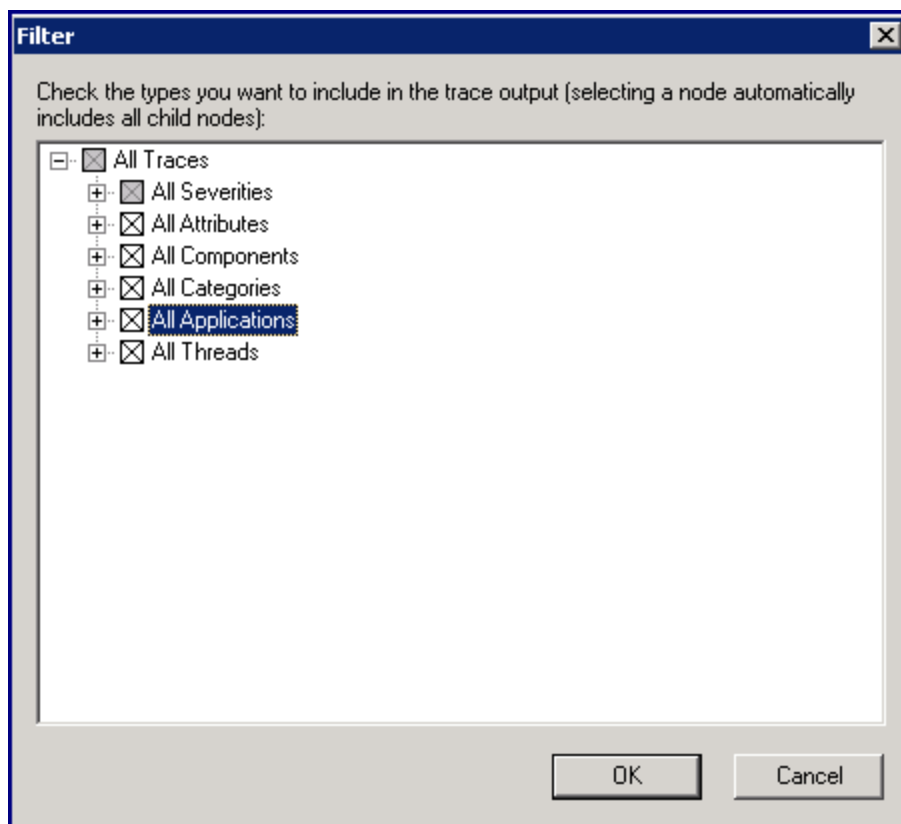
Filter Traces

The ovtrcgui utility displays all the trace messages that are logged into the trace output files based on the configuration set in the trace configuration file. You can filter the available messages to display only the messages of your choice in the ovtrcgui console. To filter the available trace messages, follow these steps:

1. In the ovtcgui console, click **View** → **Filter**. The Filter dialog box opens.



2. Expand **All Traces**. The dialog box lists all filtering parameters in the form of a tree.

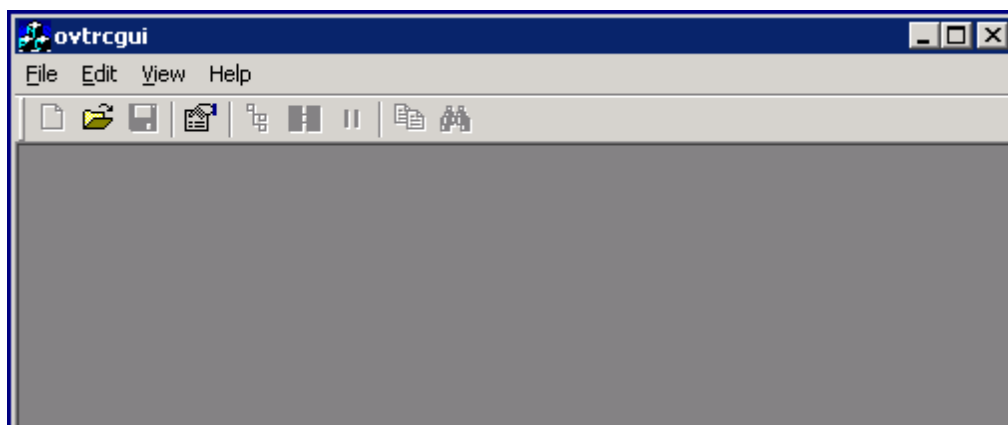


3. Expand the parameters to make selections to filter the trace messages.
4. Click **OK**. You can see only the filtered messages in the ovtrcgui console.

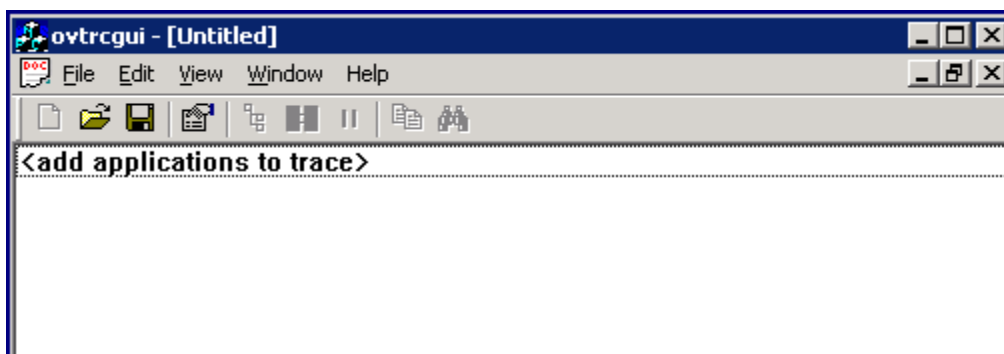
Using the Tracing GUI

On the Windows nodes, you can use the tracing GUI (the `ovtrcgui` utility) to create the trace configuration file. To use this utility and create a trace configuration file, follow these steps:

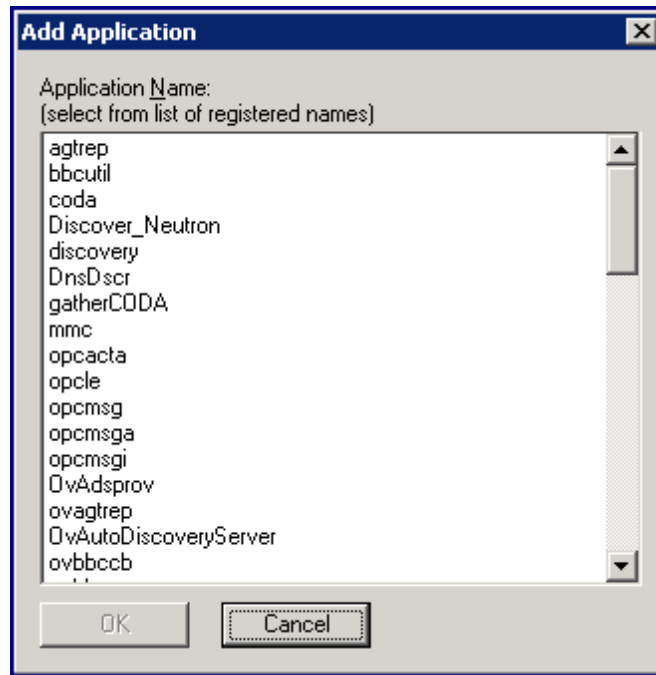
1. Run the `ovtrcgui.exe` file from the `%OvInstallDir%\support` directory. The `ovtrcgui` window opens.



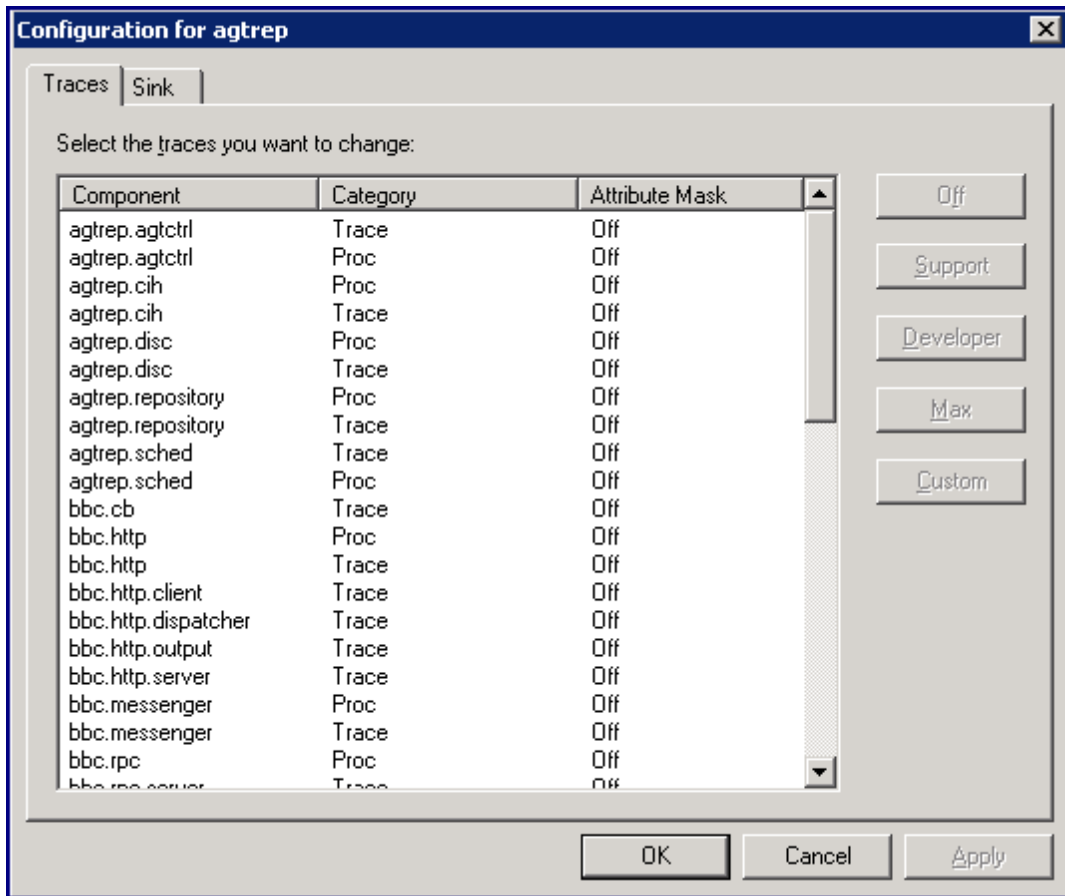
2. In the ovtrcgui window, click **File** → **New** → **Trace Configuration**. A new trace configuration editor opens.



3. In the ovtrcgui window, click **Edit** → **Add Application**. Alternatively, right-click on the editor, and then click **Add Application**. The Add Application window opens.

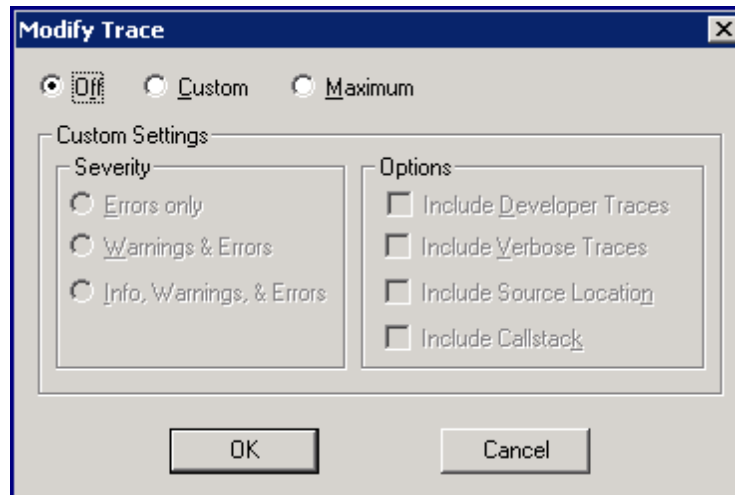


4. Select the application that you want to trace, and then click **OK**. The Configuration for *<application>* window opens.



The Traces tab of the Configuration for <application> window lists all the components and categories of the selected application. By default, tracing for all the components and categories are set to Off.

5. In the Traces tab, click a component and category pair, and then click one of the following buttons:
 - **Support:** Click this to gather trace messages marked as informational notifications.
 - **Developer:** Click this to gather trace messages marked as informational notifications along with all developer traces.
 - **Max:** Click this to set the maximum level of tracing.
 - **Custom:** When you click Custom, the Modify Trace window opens.



In the Modify Trace window, select the custom options, select the trace levels and options of your choice, and then click **OK**.

Tip: In the Configuration for <application> window, you can click **Off** to disable tracing for a component-category pair.

6. Click **OK**.
7. Go to the Sink tab.
8. Specify the name of the trace output file in the File Name text box. The file extension must be .trc.

Tip: Specify the complete path for the .trc file.

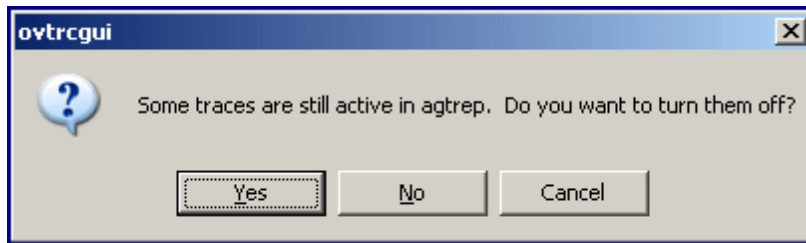
9. Specify the number of historic files from the drop-down list.
10. Specify the maximum file size from the drop-down list.
11. Click **Apply**.
12. Click **OK**.

Note: The `ovtrcgui` utility enables the tracing mechanism when you click **OK**.

13. Click **File** → **Save**. The Save As dialog box opens.

In the Save As dialog box, browse to a suitable location, specify the trace configuration file name with the .tcf extension in the File name text box, and then click **Save**.

14. The `ovtrcgui` utility saves the new trace configuration file into the specified location with the specified name and enables the tracing mechanism based on the configuration specified in the file. You can open the trace configuration file with the `ovtrcgui` utility and add new configuration details.
15. If you try to close the trace configuration editor or the `ovtrcgui` window, the following message appears:



16. If you click **No**, the tracing mechanism continues to trace the configured applications on the system. If you click **Yes**, the `ovtrcgui` utility immediately disables the tracing mechanism.

Chapter 28: Troubleshooting

This section describes the solutions or workarounds for the common problems encountered while working with the HP Operations agent. Areas covered in this section include:

- Operations Monitoring Component
- Performance Collection Component
- RTMA

Operations Monitoring Component

- **Problem:** If HPOM manages a large number of nodes (more than 1024), you may experience communication problems between HPOM and managed nodes. You can also see this problem when the HP Operations agent is installed on the HP Performance Manager server that communicates with a large number of managed nodes (more than 1024).

Solution:

To avoid this problem, go to the management server (if HPOM manages more than 1024 nodes) or the HP Performance Manager server (if the agent is installed on an HP Performance Manager server that communicates with more than 1024 nodes), and then perform the following configuration steps:

Log on as root or administrator.

Run the following command:

On Windows

```
%ovinstalldir%bin\ovconfchg -ns xpl.net -set SocketPoll true
```

On UNIX/Linux

```
/opt/OV/bin/ovconfchg -ns xpl.net -set SocketPoll true
```

Restart the agent:

On Windows

```
%ovinstalldir%bin\opcagt -stop
```

```
%ovinstalldir%bin\opcagt -start
```

On HP-UX, Linux, or Solaris:

```
/opt/OV/bin/opcagt -stop
```

```
/opt/OV/bin/opcagt -start
```

- **Problem:** On the Windows Server 2008 node, the `opcmsga` process does not function, and the `ovc` command shows the status of the `opcmsga` process as aborted.

Solution:

Set the `OPC_RPC_ONLY` variable to `TRUE` by running the following command:

```
ovconfchg -ns eaagt -set OPC_RPC_ONLY TRUE
```

- **Problem:** On Windows nodes, Perl scripts do not work from the policies.
Cause: Perl scripts available within the policies require the PATH configuration variable to include the directory where Perl (supplied with the HP Operations agent) is available.

Solution:

- a. Run the following command to set the PATH configuration variable to the Perl directory:
`ovconfchg -ns ctrl.env -set PATH "%ovinstalldir%nonOV\perl\bin"`
 - b. Restart the agent by running the following commands:
 - i. `ovc -kill`
 - ii. `ovc -start`
- **Problem:** Changes do not take effect after changing the variable values through the `ovconfchg` command.

Cause 1:

The variable requires the agent to be restarted.

Solution 1:

Restart the agent by running the following commands:

- a. `ovc -kill`
- b. `ovc -start`

Cause 2:

ConfigFile policies deployed on the node sets the variable to a certain value.

Solution 2:

If the deployed ConfigFile policies include commands to set the configuration variables to certain values, your changes through the `ovconfchg` command will not take effect. You must either remove the ConfigFile policies from the node, or modify the policies to include the commands that set the variables to the desired values.

Cause 3:

The profile or job file available on the node override your changes.

Solution 3: Open the profile or job file on the node and make sure they do not include conflicting settings for the variables.

- **Problem:** When a particular Message Interceptor (`msgi`) policy is deployed and a message that matches the condition in this policy is triggered, the `opcmsgi` process stops responding with high CPU utilization (it consumes 25 percent on 4-CPU's server, so 100 percent of a single CPU) and it does not process anymore messages.

Cause:

The `opcmsgi` process stops responding and does not process anymore messages because of the following pattern:

```
<*><@.variable><*>
```

Solution:

Replace the `<*><@.variable><*>` pattern with `<*><@.variable><S><*>`.

For example:

Original pattern:

```
^\tM_Type: <@.SiSMonType><*>SiS: <  
[http://<@.SiSIP>:<@>].SiSURL<*>Monitor:<@.SiSMonName><*>Group:  
<@.SiSGrpName><*>Status: <@.SiSMonStatus><*>URL:<@.SiSMonURL><*>Match  
Content:<@.SiSMonMatch><*>Monitor Description: DOTCOM_BACKEND:{<@.OMTargetNode>}  
{<@.OMMsgGrp>}{<*.OMApp1>}{<*.OMObj>}{<*.OMSvcID>}{<@.SiSEMSCode>}{<*.OMCIs>}  
{<*.OMMsgText>}
```

Replaced Pattern:

```
^\tM_Type: <@.SiSMonType><S><*>SiS: <  
[http://<@.SiSIP>:<@>].SiSURL<S><*>Monitor:  
<@.SiSMonName><S><*>Group:<@.SiSGrpName><S><*>Status:  
<@.SiSMonStatus><S><*>URL:<@.SiSMonURL><S><*>MatchContent:  
<@.SiSMonMatch><S><*>Monitor Description:DOTCOM_BACKEND:{<@.OMTargetNode>}  
{<@.OMMsgGrp>}{<*.OMApp1>}{<*.OMObj>}{<*.OMSvcID>}{<@.SiSEMSCode>}{<*.OMCIs>}  
{<*.OMMsgText>}
```

For example:

Original Pattern:

```
^\tM_Type: <@.SiSMonType><*>SiS: <  
[http://<@.SiSIP>:<@>].SiSURL<*>Monitor:<@.SiSMonName><*>Group:  
<@.SiSGrpName><*>Status: <@.SiSMonStatus><*>URL:<@.SiSMonURL><*>Match  
Content:<@.SiSMonMatch><*>Monitor Description: DOTCOM_AUTONOMY:{<@.OMTargetNode>}  
{<@.OMMsgGrp>}{<*.OMApp1>}{<*.OMObj>}{<*.OMSvcID>}{<@.SiSEMSCode>}{<*.OMCIs>}  
{<*.OMMsgText>}
```

Replaced Pattern:

```
^\tM_Type: <@.SiSMonType><S><*>SiS: <  
[http://<@.SiSIP>:<@>].SiSURL<S><*>Monitor:  
<@.SiSMonName><S><*>Group:<@.SiSGrpName><S><*>Status:  
<@.SiSMonStatus><S><*>URL:<@.SiSMonURL><S><*>MatchContent:  
<@.SiSMonMatch><S><*>Monitor Description:DOTCOM_AUTONOMY:{<@.OMTargetNode>}  
{<@.OMMsgGrp>}{<*.OMApp1>}{<*.OMObj>}{<*.OMSvcID>}{<@.SiSEMSCode>}{<*.OMCIs>}  
{<*.OMMsgText>}
```

- **Problem:** On HP Operations Agent, pattern matching reports wrong results.

Cause:

On HP Operations Agent, pattern matching may report wrong results if there are more than 148 OR (|) operators in the pattern.

Note: In pattern matching, OR (|) operator is a special character that separates two expressions by matching the string in both the expressions.

For example:

[ab|c]d

In the above pattern, the OR <|> operator matches the string **abd** and the string **cd**.

Solution:

Ensure that the number of OR (|) operators does not exceed 148.

- **Problem:** After changing the value of the configuration variable `SNMP_SESSION_MODE`, the status of the `opctrapi` process is shown as Aborted by `ovc`.

Cause:

After you change the value of the configuration variable `SNMP_SESSION_MODE`, the HP Operations agent attempts to restart `opctrapi`. Occasionally, the process of restarting `opctrapi` fails.

Solution:

Restart `opctrapi` by running the following command:

```
ovc -start opctrapi
```

- **Problem:** The `opcmona` process is automatically restarted after you run a schedule task policy with an embedded perl script on the node and the following message appears in the HPOM console:

```
(ctrl-208) Component 'opcmona' with pid 6976 exited with exit value '-1073741819'. Restarting component.
```

Cause:

References of `exit (o)` in the embedded perl script cause `opcmona` to restart.

Solution:

Do not use `exit (o)` in the embedded perl script.

- **Problem:** On UNIX and Linux systems, the HP Operations agent 11.15 stops responding when you stop the agent services.

Solution:

To resolve this issue, terminate the agent services using the `ovc -kill` command. The `ovc - kill` command terminates the agent services after a timeout of 40 seconds.

- **Problem:** On Red Hat Enterprise Linux system, `ovbbccb` cannot access the time zone file and writes the date information in UTC(GMT) format rather than actual time zone set for the system. This wrong time zone information is logged in the `System.txt` file.

Solution:

To establish the correct time zone, follow these steps:

- a. Run the following command to verify the location of the `zoneinfo` file:

```
ls /usr/share/zoneinfo/
```

- b. Run the following command to verify the location of the `localtime` file:

```
ls -l /etc/localtime
```

Note: Local time determines the current time zone if no "TZ" environment variable is set.

- c. Create the following directory:

```
mkdir -p /var/opt/OV/usr/share/lib
```

- d. Copy the files recursively from one folder to another:

```
cp -rp /usr/share/zoneinfo /var/opt/OV/usr/share/lib
```

- e. Create the directory named `etc` under `/var/opt/OV/` by running the following command:

```
mkdir /var/opt/OV/etc
```

- f. Copy the `localtime` file into the directory:

```
cp /etc/localtime /var/opt/OV/etc
```

- g. Restart the agent by running the following commands:

- A. `ovc -kill`
- B. `ovc -start`

Note: `ovbbccb` uses the current time zone when it later records an entry into the `System.txt` file.

Performance Collection Component

- **Problem:** The following error appears in the `status.midaemon` file on the HP-UX 11.11 system:
`mi_shared - MI initialization failed (status 28)`
Cause: Large page size of the `midaemon` binary.
Solution: To resolve this, follow these steps:
 - a. Log on to the system as the root user.
 - b. Run the following command to stop the HP Operations agent:
`/opt/OV/bin/opcagt -stop`
 - c. Run the following command to take a backup of `midaemon`:
`cp /opt/perf/bin/midaemon /opt/perf/bin/midaemon.backup`
 - d. Run the following command to reduce the page size to 4K for the `midaemon` binary:
`chattr +pi 4K /opt/perf/bin/midaemon`
 - e. Run the following command to start the HP Operations agent:
`/opt/OV/bin/opcagt -start`
- **Problem:** After installing the HP Operations agent, the following error message appears in the `System.txt` file if the tracing mechanism is enabled:
`Scope data source initialization failed`
Solution: Ignore this error.
- **Problem:** The following error message appears in the HPOM console:
`CODA: GetDataMatrix returned 76='Method ScopeDataView::CreateViewEntity failed`
Cause: This message appears if you use the `PROCESS` object with the `SCOPE` data source in Measurement Threshold policies where the source is set to Embedded Performance Component.
Solution: Use the `service/process` monitoring policy instead.
- **Problem:** When scope (`scopeux` on UNIX nodes and `scopent` on windows nodes) reads and checks records during startup or roll over there may be failure in reading records and the following error message appears in the **status.scope** file:
`ERROR: Unable to read from 'C:\Program Files\HP Openview\data\datafiles\logappl'
- corrupted data (-793044). (PE221-24)`
`**** C:\Program : 02/16/15 11:20:54 ****`
`WARNING: Unable to lock record in log file. (PE221-48)`
Cause: When log files are corrupted, Scope does not identify the corrupted log files but continues to log data. Scope identifies the corrupted log files only during roll over.
Solution:

You must set the **SCOPESTARTSCAN** environment variable for scope. If this variable is set to TRUE, and if the log files are corrupted, scope does a backup of the corrupted log files and re-creates them.

Note: Make sure that no other process is reading the scope log files during scope startup because the process can lock the log files when scope re-creates them.

For example:

- When CODA process reads scope log files, make sure you stop coda using the `ovc -stop coda` command.
- When extract and utility tools read SCOPE log files, make sure you stop the extract and utility programs.

- *Problem:* Data analysis products, such as HP Performance Manager or HP Service Health Reporter, fail to retrieve data from agent's data store and show the following error:

```
Error occurred while retrieving data from the data source
```

Cause: The data access utility of the agent reads all the records of a data class against a single query from a client program. Queries are sent to the agent by data analysis clients like HP Performance Manager. When a data class contains a high volume of records, the data access utility fails to process the query.

Solution: To avoid this issue, configure the data access utility to transfer data records to the client in multiple chunks. Follow these steps:

- a. Log on to the agent node with as root or administrator.
- b. Run the following command:

On Windows:

```
%ovinstalldir%\bin\ovconfchg -ns coda -set DATAMATRIX_VERSION 1
```

On HP-UX, Solaris, or Linux:

```
/opt/OV/bin/ovconfchg -ns coda -set DATAMATRIX_VERSION 1
```

On AIX:

```
/usr/lpp/OV/bin/ovconfchg -ns coda -set DATAMATRIX_VERSION 1
```

For each query, the agent's data access utility now breaks the data into chunks of five records, and then sends the data to the client program. Breaking the data into chunks enhances the performance of data transfer process.

You can control the number of records that the agent can send to the client with every chunk. The `DATAMATRIX_ROWCOUNT` variable (available under the `coda` namespace) enables you to control this number (the default value is five).

Decreasing the value of the `DATAMATRIX_ROWCOUNT` variable may marginally increase the data transfer rate when you have very large volumes of data into the data store.

When the `DATAMATRIX_ROWCOUNT` variable is set to 0, the HP Operations agent reverts to its default behavior of sending data records without chunking.

However, it is recommended that you do not change the default setting of the `DATAMATRIX_ROWCOUNT` variable.

- c. Restart the agent for the changes to take effect.

```
ovc -restart coda
```
- **Problem:** On Red Hat Enterprise Linux system when the number of file descriptors exceed the limit, the ARM applications delay by 10 seconds.
Solution: Set the `TIMEOUT_SEC_USEC` environment variable in the `ttd.conf` file before you trigger the ARM applications by using the following syntax:

```
export TIMEOUT_SEC_USEC=<Seconds>,<Microseconds>
```

The following "[Table 14: Different instances of configuring the TIMEOUT_SEC_USEC environment variable](#)" below lists the different instances of configuring the `TIMEOUT_SEC_USEC` environment variable:

Table 14: Different instances of configuring the TIMEOUT_SEC_USEC environment variable

<code>TIMEOUT_SEC_USEC=<Seconds>,</code>	<p>If the value of <i>Microseconds</i> is not mentioned, the default timeout of 0 microseconds is considered.</p> <p>For such instances where the <i>Microseconds</i> parameter is not mentioned, add a comma after the <i>Seconds</i> parameter.</p>
<code>TIMEOUT_SEC_USEC=,<Microseconds></code>	<p>If the value of <i>Seconds</i> is not mentioned, the default timeout of 0 seconds is considered.</p> <p>For such instances where the <i>Seconds</i> parameter is not mentioned, add a comma before the <i>Microseconds</i> parameter.</p>
<code>TIMEOUT_SEC_USEC=</code>	<p>If the values of both the <i>Seconds</i> and <i>Microseconds</i> parameter are not mentioned, the default timeout values of 10 seconds and 0 microseconds are considered.</p>
<code>TIMEOUT_SEC_USEC=<Seconds>,<Microseconds></code>	<p>If the values of both the <i>Seconds</i> and <i>Microseconds</i> parameter are mentioned, then the values mentioned will replace the default values of 10 seconds and 0 microseconds.</p>

Note: If the parameter values are configured to a lowest value, for example, 100 milliseconds, you may lose the transaction records.

RTMA

- **Problem:** On the vSphere Management Assistant (vMA) node, the `rtmd` process does not function, and the `ovc` command shows the status of the `rtmd` process as aborted.
Cause: The `rtmd` process cannot resolve the hostname of the system to the IP address.
Solution:
 - a. Log on to the node with the root privileges.
 - b. From the `/etc` directory, open the `hosts` file with a text editor.
 - c. Locate the line where the term `localhost` appears.
 - d. Remove the `#` character from the beginning of the line.
 - e. Save the file.

- f. Start all processes by running the following command:

```
ovc -restart
```

- **Problem:** The Diagnostic view of HP Performance Manager cannot access data.

Cause: The `rtmd` process is not running.

Solution: To check if the `rtmd` process is running on the HP Operations agent node, run `ovc -status rtmd`. To start the `rtmd` process, run `ovc -start rtmd`.

- **Problem:** The following error appears in the `status.perfd` file on the HP-UX 11.11 system:
`mi_shared - MI initialization failed (status 28)`

Cause: Large page size of the `perfd` binary.

Solution: To resolve this, follow these steps:

- a. Log on to the system as the root user.
- b. Run the following command to stop the HP Operations agent:
`/opt/OV/bin/opcagt -stop`
- c. Run the following command to take a backup of `perfd`:
`cp /opt/perf/bin/perfd /opt/perf/bin/perfd.backup`
- d. Run the following command to reduce the page size to 4K for the `perfd` binary:
`chattr +pi 4K /opt/perf/bin/perfd`
- e. Run the following command to start the HP Operations agent:
`/opt/OV/bin/opcagt -start`

HP GlancePlus

Problem: HP GlancePlus does not show all LPAR instances hosted on an AIX frame.

Cause: The LPAR where you installed the HP Operations agent cannot communicate with other LPARs hosted on the AIX frame.

Solution: Make sure that the LPAR where you installed the HP Operations agent can communicate with all other LPARs hosted on the AIX frame.

Run the following command on the LPAR that hosts the HP Operations agent to check its connectivity with other LPARs:

```
xmpeek -l <hostname>
```

In this instance, `<hostname>` is the host name of an LPAR.

Send Documentation Feedback

If you have comments about this document, you can [contact the documentation team](#) by email. If an email client is configured on this system, click the link above and an email window opens with the following information in the subject line:

Feedback on User Guide (Operations Agent 11.16)

Just add your feedback to the email and click send.

If no email client is available, copy the information above to a new message in a web mail client, and send your feedback to docfeedback@hpe.com.

We appreciate your feedback!