HP LoadRunner

for the Windows operating systems

Software Version: 9.10

![separator bar]

Controller User's Guide

hp ®

i n v e n t

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

Third-Party Web Sites

HP provides links to external third-party Web sites to help you find supplemental information. Site content and availability may change without notice. HP makes no representations or warranties whatsoever as to site content or availability.

Copyright Notices

Trademark Notices

Java™ is a US trademark of Sun Microsystems, Inc.

Microsoft® and Windows® are U.S. registered trademarks of Microsoft Corporation.

Oracle® is a registered US trademark of Oracle Corporation, Redwood City, California.

UNIX® is a registered trademark of The Open Group.

Documentation Updates

This manual's title page 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:

http://h20230.www2.hp.com/selfsolve/manuals

Support

You can visit the HP Software Support Web site at: **www.hp.com/go/hpsoftwaresupport**

HP Software online support provides an efficient way to access interactive technical support tools. As a valued support customer, you can benefit by using the support 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 find more information about access levels, go to: **http://h20230.www2.hp.com/new_access_levels.jsp**

To register for an HP Passport ID, go to: **http://h20229.www2.hp.com/passport-registration.html**

Table of Contents

PART II: DESIGNING A SCENARIO

PART V: WORKING WITH DIAGNOSTICS

PART VI: MONITORING A SCENARIO

Table of Contents

Welcome to This Guide

Welcome to the *HP LoadRunner Controller User's Guide*. This guide presents an overview of the HP LoadRunner testing process, and describes how to create and run HP LoadRunner scenarios using HP LoadRunner Controller in a Windows environment.

HP LoadRunner is HP's tool for application performance testing. LoadRunner stresses your entire application to isolate and identify potential client, network, and server bottlenecks.

LoadRunner enables you to test your system under controlled and peak load conditions. To generate load, LoadRunner runs thousands of Virtual Users, or **Vusers**, that are distributed over a network. The Vusers can run on UNIX and Windows-based platforms. Using a minimum of hardware resources, these Vusers provide consistent, repeatable, and measurable load to exercise your application just as real users would. LoadRunner's in–depth reports and graphs provide the information that you need to evaluate the performance of your application.

This chapter includes:

➤ How This Guide Is Organized on page 14

➤ Who Should Read This Guide on page 15

➤ LoadRunner Online Documentation on page 15

➤ Additional Online Resources on page 16

➤ Typographical Conventions on page 17

How This Guide Is Organized

This guide contains the following parts:

Part I   Understanding LoadRunner

Introduces HP LoadRunner Controller, and describes how you use it for load testing.

Part II   Designing a Scenario

Explains how to create a scenario for load testing.

Part III   Executing a Scenario

Explains how to run a scenario, and how to view the system's performance during the scenario.

Part IV   Working with Firewalls

Explains how to use HP LoadRunner Controller in an environment that includes firewalls.

Part V   Working with Diagnostics

Explains how to use LoadRunner's Diagnostics modules to identify and pinpoint performance problems in Siebel, Oracle, SAP, J2EE, and .NET environments.

Part VI   Monitoring a Scenario

Explains how to monitor scenario execution using the LoadRunner online monitors.

Part VII   Appendixes

Contains additional information about using LoadRunner.

Who Should Read This Guide

This guide is for the following users of HP LoadRunner:

➤ Performance Engineers

➤ Project Manager

This document assumes that you are moderately knowledgeable about enterprise application development and highly skilled in enterprise system and database administration.

LoadRunner Online Documentation

LoadRunner includes the following online documentation:

➤ **Readme.** Provides last-minute news and information about LoadRunner. You access the Readme from the **Start** menu.

➤ **Books Online/Printer-Friendly Documentation.** Includes PDF versions of the guides. Select **Help** > **Books Online**.

➤ **Online Help.** Available from specific LoadRunner windows by clicking in the window and pressing **F1** or clicking the **Help** button.

➤ **LoadRunner Online Help** includes:

➤ **Error Codes Troubleshooting.** Provides clear explanations and troubleshooting tips for Controller connectivity and Web protocol errors. It also provides general troubleshooting tips for Winsock, SAPGUI, and Citrix protocols.

➤ **LoadRunner Agent Configuration Tool Online Help.** Provides help on the Agent Configuration Tool, accessed by clicking the **Help** button in the Agent Configuration dialog box (**Start** > **LoadRunner** > **Advanced Settings** > **Agent Configuration**).

➤ **LoadRunner Controller Automation COM and Monitor Automation Reference.** An interface with which you can write programs to run the LoadRunner Controller and perform most of the actions available in the Controller user interface. You access the reference from the LoadRunner online documentation.

➤ **LoadRunner Function Reference.** Gives you online access to all of LoadRunner's functions that you can use when creating Vuser scripts, including examples of how to use the functions. Check HP's Customer Support Web site for updates to the *HP LoadRunner Online Function Reference*.

Additional Online Resources

HP Software Self-solve knowledge base. This link enables you to browse the HP Software Self-solve knowledge base and add your own articles. The URL for this Web site is http://h20230.www2.hp.com/selfsolve/documents.

HP Software Support Web site. This site enables you to access the HP Software Self-solve knowledge base, post to and search user discussion forums, submit support requests, download patches and updated documentation, and more. The URL for this Web site is http://www.hp.com/go/hpsoftwaresupport. You can also access this Web site from the **Help** menu.

HP Software Web site. This site provides you with the most up-to-date information on HP Software products. This includes new software releases, seminars and trade shows, customer support, and more. The URL for this Web site is www.hp.com/go/software. You can also access this Web site from the **Help** menu.

Typographical Conventions

This guide uses the following typographical conventions:

UI Elements and **Function Names**	This style indicates the names of interface elements on which you perform actions, file names or paths, and other items that require emphasis. For example, "Click the **Save** button." It also indicates method or function names. For example, "The **wait_window** statement has the following parameters:"
Arguments	This style indicates method, property, or function arguments and book titles. For example, "Refer to the *HP User's Guide*."
<**Replace Value**>	Angle brackets enclose a part of a file path or URL address that should be replaced with an actual value. For example, <**MyProduct installation folder**>**bin**.
Example	This style is used for examples and text that is to be typed literally. For example, "Type Hello in the edit box."
CTRL+C	This style indicates keyboard keys. For example, "Press ENTER."
[ ]	Square brackets enclose optional arguments.
{ }	Curly brackets indicate that one of the enclosed values must be assigned to the current argument.
...	In a line of syntax, an ellipsis indicates that more items of the same format may be included. In a programming example, an ellipsis is used to indicate lines of a program that were intentionally omitted.
		A vertical bar indicates that one of the options separated by the bar should be selected.

Part I

Understanding LoadRunner

1

Introduction

HP LoadRunner load tests your application by emulating an environment in which multiple users work concurrently. While the application is under load, LoadRunner accurately measures, monitors, and analyzes a system's performance and functionality.

This chapter includes:

Application Load Testing

Modern system architectures are complex. While they provide an unprecedented degree of power and flexibility, these systems are difficult to test. Whereas single-user testing focuses primarily on functionality and the user interface of a system component, application testing focuses on performance and reliability of an entire system.

For example, a typical application testing scenario might depict 1000 users that log in simultaneously to a system on Monday morning. What is the response time of the system? Does the system crash? To be able to answer these questions—and more—a complete application performance testing solution must:

➤ test a system that combines a variety of software applications and hardware platforms

➤ determine the suitability of a server for any given application

➤ test the server before the necessary client software has been developed

➤ emulate an environment where multiple clients interact with a single server application

➤ test an application under the load of tens, hundreds, or even thousands of potential users

Manual Testing Limitations

Traditional or manual testing methods offer only a partial solution to load testing. For example, you can test an entire system manually by constructing an environment where many users work simultaneously on the system. Each user works at a single machine and submits input to the system. However, this manual testing method has the following drawbacks:

➤ It is expensive, requiring large amounts of both personnel and machinery.

➤ It is complicated, especially coordinating and synchronizing multiple testers.

➤ It involves a high degree of organization, especially to record and analyze results meaningfully.

➤ The repeatability of the manual tests is limited.

The HP LoadRunner Solution

LoadRunner addresses the drawbacks of manual performance testing:

➤ LoadRunner reduces personnel requirements by replacing human users with virtual users or **Vusers**. These Vusers emulate the behavior of real users— operating real applications.

➤ Because numerous Vusers can run on a single computer, LoadRunner reduces the amount of hardware required for testing.

➤ The HP LoadRunner Controller allows you to easily and effectively control all the Vusers—from a single point of control.

➤ LoadRunner monitors the application performance online, enabling you to fine-tune your system during test execution.

➤ LoadRunner automatically records the performance of the application during a test. You can choose from a wide variety of graphs and reports to view the performance data.

➤ LoadRunner checks where performance delays occur: network or client delays, CPU performance, I/O delays, database locking, or other issues at the database server. LoadRunner monitors the network and server resources to help you improve performance.

➤ Because LoadRunner tests are fully automated, you can easily repeat them as often as you need.

Using HP LoadRunner

➤ **Scenario.** Using HP LoadRunner, you divide your application performance testing requirements into **scenarios**. A scenario defines the events that occur during each testing session. Thus, for example, a scenario defines and controls the number of users to emulate, the actions that they perform, and the machines on which they run their emulations.

➤ **Vusers.** In the scenario, LoadRunner replaces human users with **virtual users** or **Vusers**. When you run a scenario, Vusers emulate the actions of human users working with your application. While a workstation accommodates only a single human user, many Vusers can run concurrently on a single workstation. In fact, a scenario can contain tens, hundreds, or even thousands of Vusers.

➤ **Vuser Scripts.** The actions that a Vuser performs during the scenario are described in a Vuser script. When you run a scenario, each Vuser executes a **Vuser script**. The Vuser scripts include functions that measure and record the performance of your application's components.

➤ **Transactions.** To measure the performance of the server, you define **transactions**. A transaction represents an action or a set of actions that you are interested in measuring. You define transactions within your Vuser script by enclosing the appropriate sections of the script with **start** and **end** transaction statements. For example, you can define a transaction that measures the time it takes for the server to process a request to view the balance of an account and for the information to be displayed at the ATM.

➤ **Rendezvous points**. You insert **rendezvous points** into Vuser scripts to emulate heavy user load on the server. **Rendezvous points** instruct Vusers to wait during test execution for multiple Vusers to arrive at a certain point, so that they may simultaneously perform a task. For example, to emulate peak load on the bank server, you can insert a rendezvous point instructing 100 Vusers to deposit cash into their accounts at the same time.

➤ **Controller.** You use the **HP LoadRunner Controller** to manage and maintain your scenarios. Using the Controller, you control all the Vusers in a scenario from a single workstation.

➤ **Load Generator.** When you execute a scenario, the Controller distributes each Vuser in the scenario to a **load generator**. The load generator is the machine that executes the Vuser script, enabling the Vuser to emulate the actions of a human user.

➤ **Performance analysis.** Vuser scripts include functions that measure and record system performance during load-testing sessions. During a scenario run, you can monitor the network and server resources. Following a scenario run, you can view **performance analysis** data in reports and graphs.

Working with HP LoadRunner

Suppose you want to test an online banking Web server that is accessed by many Internet users. The Web site provides a full range of banking services to the customers—such as the ability to transfer funds and check account balances. To test this server, you create a scenario. The scenario defines the actions that are performed on the server during the load test.

During the scenario that loads and monitors the bank server, you want to:

➤ emulate conditions of controlled load on the server

➤ emulate conditions of maximum load on the server

➤ measure server performance under load

➤ check where performance delays occur: network or client delays, CPU performance, I/O delays, database locking, or other issues at the server

➤ monitor the network and server resources under load

HP LoadRunner Vuser Technology

On each Windows load generator, you install the **Remote Agent Dispatcher (Process)** and a LoadRunner **Agent**.

Controller
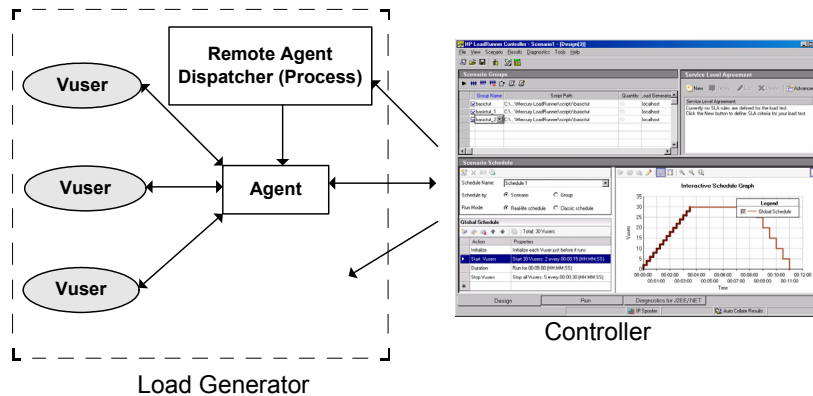
Load Generator

➤ **Remote Agent Dispatcher.** The Remote Agent Dispatcher (Process) enables the Controller to start applications on the load generator.

➤ **Agent.** The LoadRunner Agent enables the Controller and the load generator to communicate with each other. When you run a scenario, the Controller instructs the Remote Agent Dispatcher (Process) to launch the LoadRunner agent. The agent receives instructions from the Controller to initialize, run, pause, and stop Vusers. At the same time, the agent also relays data on the status of the Vusers back to the Controller.

HP LoadRunner Vuser Types

LoadRunner supports various types of Vusers using the most common protocols. Each type is designed to handle different aspects of today's system architectures. You can use the Vuser types in any combination in a scenario in order to create a comprehensive application test.

Vuser types are divided into several categories, indicating their use within common business envrionments. For example:

➤ **Application Deployment Solution** Vusers support the Citrix or Remote Desktop Protocol (RDP) protocols.

➤ **Client/Server** Vusers support database and socket communication protocols.

➤ **Distributed Components** Vusers support COM/DCOM and Microsoft .NET protocols.

➤ **E-business** Vusers support AJAX, AMF, HTTP/HTML, and Web Services protocols.

➤ **ERP/CRM** Vusers provide testing for Oracle NCA, SAP, and Siebel systems .

➤ **Java** Vusers support the Jacada, CORBA, RMI, and JMS protocols.

For a complete list of the avaialbe protocols, refer to the *HP Virtual User Generator User's Guide*.

GUI Vusers

LoadRunner can integrate functional testing scripts in the form of *GUI Vuser scripts* into a load testing scenario. These GUI Vuser scripts are created using HP Functional Testing software - QuickTest Professional or WinRunner.

You can run only a single GUI Vuser on a Windows-based load generator. Use Citrix to run multiple GUI Vusers. For additional information on Windows-based GUI Vusers, see Chapter 14, "Using Functional Testing Scripts in LoadRunner."

Vuser Technology

Vusers (except for GUI Vusers) generate load on a server by submitting input directly to the server. Vusers do not operate client applications—they access the server using LoadRunner API functions. These API functions emulate the input from an actual application.
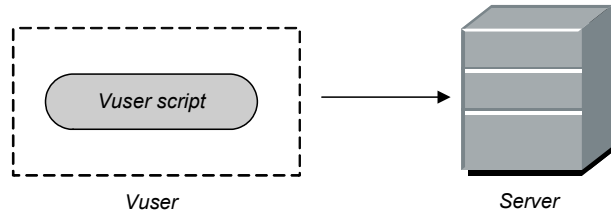
Vuser   *Server*

Because Vusers are not reliant on client software, you can use Vusers to test server performance even before the client software has been developed. Further, since Vusers do not have a user interface, the amount of system resources required is minimal. This allows you to run large numbers of Vusers on a single workstation.

The following example illustrates the use of Vusers: Suppose that you have a Web-based database server that maintains your customer information. The information is accessed by numerous customer service personnel who are located throughout the country. The server receives the queries, processes the requests, and returns responses via the Web to field personnel.

You want to test the response times of the entire system when numerous service personnel simultaneously access the server. Using LoadRunner, you could create several hundred Vusers, each Vuser accessing the server database. The Vusers enable you to emulate and measure the performance of your database and Web servers under the load of many users.

You develop a Vuser script to define the actions of a Vuser. A Vuser script includes functions that control the script execution, specify the input that the Vuser submits to the server, and measure the server performance.

You develop Vuser scripts either by recording with HP Virtual User Generator (VuGen) or by using LoadRunner's Vuser script templates.

For the database server example above, you could create a Vuser script that performs the following actions:

➤ Logs in to the Web application

➤ Connects to the database server

➤ Submits an SQL query

➤ Retrieves and processes the server response

➤ Disconnects from the server and the Web

You can create Vuser scripts on a Windows-based platform, or program them on a UNIX platform. For a list of the supported UNIX platforms, see the HP LoadRunner Readme file. For more information about Vusers, see the *HP Virtual User Generator User's Guide*.

2

The LoadRunner Testing Process

You can easily create and run load test scenarios by following the LoadRunner testing process below.

This chapter gives you an overview of LoadRunner's six–step process for testing your Web-based application under load.

The following illustration outlines the testing process

Step I	Planning the Test
Step II	Creating Vuser Scripts
Step III	Creating the Scenario
Step IV	Running the Scenario
Step V	Monitoring the Scenario
Step VI	Analyzing Test Results
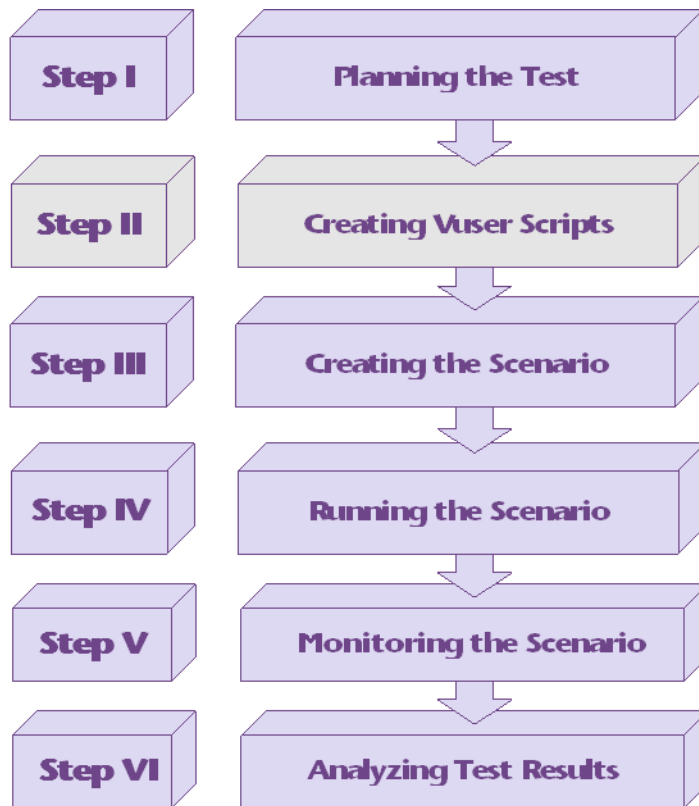
Step I: Planning the Test

Successful load testing requires that you develop a thorough test plan. A clearly defined test plan will ensure that the LoadRunner scenarios that you develop will accomplish your load testing objectives. For more information, see Chapter 3, "Load Test Planning."

Step II: Creating the Vuser Scripts

Vusers emulate human users interacting with your Web-based application. A Vuser script contains the actions that each Vuser performs during scenario execution.

In each Vuser script, you determine the tasks that will be:

➤ performed by each Vuser

➤ performed simultaneously by multiple Vusers

➤ measured as transactions

For more information on creating Vuser scripts, see the *HP Virtual User Generator User's Guide*.

Step III: Creating the Scenario

A scenario describes the events that occur during a testing session. A scenario includes a list of machines on which Vusers run, a list of scripts that the Vusers run, and a specified number of Vusers or Vuser groups that run during the scenario. You create scenarios using the Controller. For an introduction to the Controller, see Chapter 4, "The Controller at a Glance."

Creating a Manual Scenario

You create a scenario by defining Vuser groups to which you assign a quantity of individual Vusers, Vuser scripts, and load generators to run the scripts. For more information, see Chapter 5, "Creating Manual Scenarios."

You can also create a scenario using the Percentage Mode, in which you define the total number of Vusers to be used in the scenario, and the load generators and percentage of the total number of Vusers to be assigned to each Vuser script. For more information, see Chapter 6, "Creating a Manual Scenario Using the Percentage Mode."

Creating a Goal-Oriented Scenario

For Web tests, you can create a goal-oriented scenario, in which you define the goals you want your test to achieve. LoadRunner automatically builds a scenario for you, based on these goals. For more information, see Chapter 9, "Creating a Goal-Oriented Scenario."

Step IV: Running the Scenario

You emulate user load on the server by instructing multiple Vusers to perform tasks simultaneously. You can set the level of load by increasing and decreasing the number of Vusers that perform tasks at the same time. For more information, see Chapter 10, "Using Rendezvous Points."

Before you run a scenario, you set the scenario configuration and scheduling. This determines how all the load generators and Vusers behave when you run the scenario. For more information, see Chapter 11, "Configuring a Scenario" and Chapter 7, "Scheduling Scenarios."

You can run the entire scenario, groups of Vusers (Vuser groups), or individual Vusers. While a scenario runs, LoadRunner measures and records the transactions that you defined in each Vuser script. You can also monitor your system's performance online. For more information, see Part III, "Executing a Scenario."

Step V: Monitoring a Scenario

You can monitor scenario execution using the LoadRunner online run-time, transaction, system resource, Web resource, Web server resource, Web application server resource, database server resource, network delay, streaming media resource, firewall server resource, ERP/CRM server resource, Java performance, J2EE & .NET Diagnostics, application deployment, middleware performance, application component, and infrastructure resources monitors. For more information, see Part VI, "Monitoring a Scenario."

Step VI: Analyzing Test Results

During scenario execution, LoadRunner records the performance of the application under different loads. You use LoadRunner's graphs and reports to analyze the application's performance. For more information about LoadRunner's reports and graphs, see the *HP LoadRunner Analysis User's Guide*.

3

Load Test Planning

Developing a comprehensive test plan is a key to successful load testing. A clearly defined test plan ensures that the LoadRunner scenarios you develop will accomplish your load testing objectives. This chapter introduces the load test planning process.

This chapter includes:

About Load Test Planning

As in any type of system testing, a well-defined test plan is the first essential step to successful testing. Planning your load testing helps you to:

➤ build test scenarios that accurately emulate your working environment.

Load testing means testing your application under typical working conditions, and checking for system performance, reliability, capacity, and so forth.

➤ understand which resources are required for testing.

Application testing requires hardware, software, and human resources. Before you begin testing, you should know which resources are available and decide how to use them effectively.

➤ define success criteria in measurable terms.

Focused testing goals and test criteria ensure successful testing. For example, it's not enough to define vague objectives like "Check server response time under heavy load." A more focused success criterion would be "Check that 50 customers can check their account balance simultaneously, and that the server response time will not exceed one minute."

Load test planning is a three-step process:
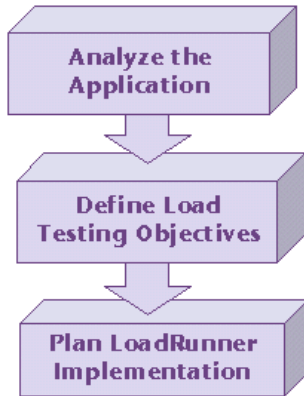
Analyzing the Application

The first step to load test planning is analyzing your application. You should become thoroughly familiar with the hardware and software components, the system configuration, and the typical usage model. This analysis ensures that the testing environment you create using LoadRunner will accurately reflect the environment and configuration of the application under test.

Identifying System Components

Draw a schematic diagram to illustrate the structure of the application. If possible, extract a schematic diagram from existing documentation. If the application under test is part of a larger network system, you should identify the component of the system to be tested. Make sure the diagram includes all system components, such as client machines, network, middleware, and servers.

The following diagram illustrates an online banking system that is accessed by many Web users. The Web users each connect to the same database to transfer funds and check balances. The customers connect to the database server through the Web, using multiple browsers.
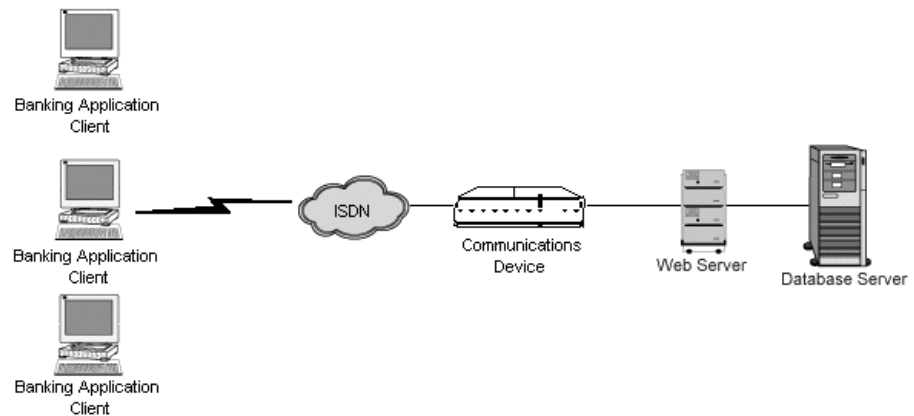
Describing the System Configuration

Enhance the schematic diagram with more specific details. Describe the configuration of each system component. You should be able to answer the following questions:

➤ How many users are anticipated to connect to the system?

➤ What is the application client's machine configuration (hardware, memory, operating system, software, development tool, and so forth)?

➤ What types of database and Web servers are used (hardware, database type, operating system, file server, and so forth)?

➤ How does the server communicate with the application client?

➤ What is the middleware configuration and application server between the front-end client and back-end server?

➤ What other network components may affect response time (modems and so forth)?

➤ What is the throughput of the communications devices? How many concurrent users can each device handle?

For example, the schematic diagram above specified that there are multiple application clients accessing the system.

Front-End Client Configuration	
Anticipated number of application clients	50 concurrent application clients
Hardware / Memory	Pentium IV/ 1GB
Operating system & version	Windows 2000
Client browser	Internet Explorer 6.0 SP1

Analyzing the Usage Model

Define how the system is typically used, and decide which functions are important to test. Consider who uses the system, the number of each type of user, and each user's common tasks. In addition, consider any background load that might affect the system response time.

For example, suppose 200 employees log on to the accounting system every morning, and the same office network has a constant background load of 50 users performing various word processing and printing tasks. You could create a LoadRunner scenario with 200 virtual users signing in to the accounting database, and check the server response time.

To check how background load affects the response time, you could run your scenario on a network where you also simulate the load of employees performing word processing and printing activities.

Task Distribution

In addition to defining the common user tasks, examine the distribution of these tasks. For example, suppose the bank uses a central database to serve clients across many states and time zones. The 250 application clients are located in two different time zones, all connecting to the same Web server. There are 150 in Chicago and 100 in Detroit. Each begins their business day at 9:00 AM, but since they are in different time zones, there should never be more than 150 users signing in at any given time. You can analyze task distribution to determine when there is peak database activity, and which activities typically occur during **peak load** time.

Defining Testing Objectives

Before you begin testing, you should define exactly what you want to accomplish.

Following are common application testing objectives that LoadRunner helps you test, as described in Robert W. Buchanan, Jr's *The Art of Testing Network Systems* (John Wiley & Sons, Inc., 1996).

Objective	Answers the Question
Measuring end-user response time	How long does it take to complete a business process?
Defining optimal hardware configuration	Which hardware configuration provides the best performance?
Checking reliability	How hard or long can the system work without errors or failures?
Checking hardware or software upgrades	How does the upgrade affect performance or reliability?
Evaluating new products	Which server hardware or software should you choose?
Measuring system capacity	How much load can the system handle without significant performance degradation?
Identifying bottlenecks	Which element is slowing down response time?

A more detailed description of each objective appears at the end of this chapter.

Stating Objectives in Measurable Terms

Once you decide on your general load testing objectives, you should identify more focused goals by stating your objectives in measurable terms. To provide a baseline for evaluation, determine exactly what constitutes acceptable and unacceptable test results.

For example:

➤ **General Objective.** Product Evaluation: choose hardware for the Web server.

➤ **Focused Objective.** Product Evaluation: run the same group of 300 virtual users on two different servers, HP and NEC. When all 300 users simultaneously browse the pages of your Web application, determine which hardware gives a better response time.

Deciding When to Test

Load testing is necessary throughout the product life cycle. The following table illustrates what types of tests are relevant for each phase of the product life cycle:

Planning and Design	Development	Deployment	Production	Evolution
Evaluate new products	Measure response time	Check reliability	Measure response time	Check HW or SW upgrades
Measure response time	Check optimal hardware configuration	Measure response time	Identify bottlenecks	Measure system capacity
	Check HW or SW upgrades	Measure system capacity		
	Check reliability			

Planning LoadRunner Implementation

The next step is to decide how to use LoadRunner to achieve your testing goals.

Defining the Scope of Performance Measurements

You can use LoadRunner to measure response time at different points in the application. Determine where to run the Vusers and which Vusers to run according to the test objectives:

➤ **Measuring end-to-end response time.** You can measure the response time that a typical user experiences by running a GUI Vuser at the front end. GUI Vusers emulate real users by submitting input to and receiving output from the client application.

You can run GUI Vusers at the front end to measure the response time across the entire network, including a terminal emulator or GUI front end, network, and server.

Client   *Middleware*   *Server*
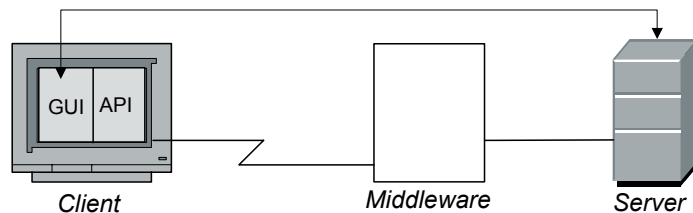
➤ **Measuring network and server response times.** You can measure network and server response time, excluding response time of the GUI front end, by running Vusers (not GUI) on the client machine. Vusers emulate client calls to the server without the user interface. When you run many Vusers from the client machine, you can measure how the load affects network and server response time.

Client   *Middleware*   *Server*
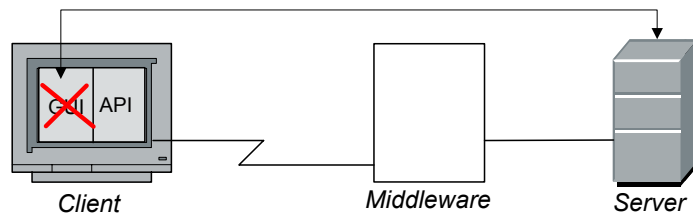
➤ **Measuring GUI response time.** You can determine how the client application interface affects response time by subtracting the previous two measurements:

GUI response time = end-to-end - network and server
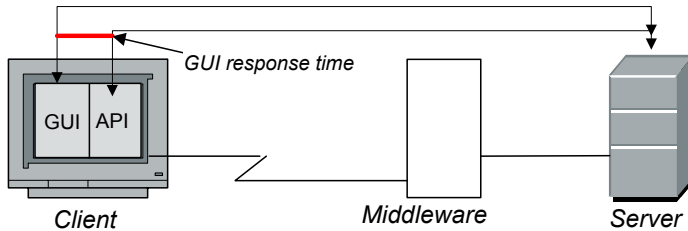
➤ **Measuring server response time.** You can measure the time it takes for the server to respond to a request without going across the network. When you run Vusers on a machine directly connected to the server, you can measure server performance.
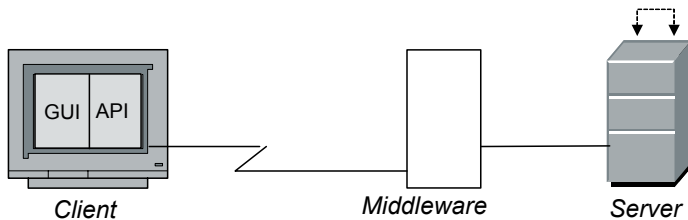
➤ **Measuring middleware-to-server response time.** You can measure response time from the server to middleware if you have access to the middleware and its API. You can create Vusers with the middleware API and measure the middleware-server performance.
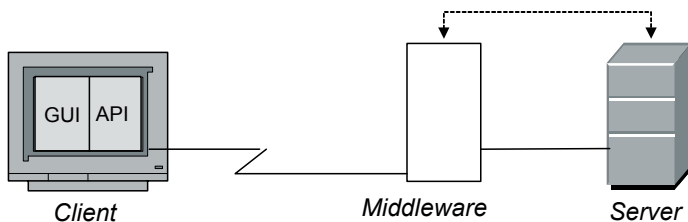
Defining Vuser Activities

Create Vuser scripts based on your analysis of Vuser types, their typical tasks and your test objectives. Since Vusers emulate the actions of a typical end-user, the Vuser scripts should include the typical end-user tasks. For example, to emulate an online banking client, you should create a Vuser script that performs typical banking tasks. You would browse the pages that you normally visit to transfer funds or check balances.

You decide which tasks to measure based on your test objectives and define **transactions** for these tasks. Transactions measure the time that it takes for the server to respond to tasks submitted by Vusers (end-to-end time). For example, to check the response time of a bank Web server supplying an account balance, define a transaction for this task in the Vuser script.

In addition, you can emulate peak activity by using **rendezvous points** in your script. Rendezvous points instruct multiple Vusers to perform tasks at exactly the same time. For example, you can define a rendezvous to emulate 70 users simultaneously updating account information.

Selecting Vusers

Before you decide on the hardware configuration to use for testing, determine the number and type of Vusers required. To decide how many Vusers and which types to run, look at the typical usage model, combined with the testing objectives. Some general guidelines are:

➤ Use one or a few GUI users to emulate each type of typical user connection.

➤ Run multiple Vusers to generate the rest of the load for each user type.

For example, suppose that you have five kinds of users, each performing a different business process:

Usage Model	GUI	Other
100 customer service users in New York (LAN connection)	2	98
30 customers in Europe (dial-in ISDN connection)	2	28

Usage Model	GUI	Other
5 background batch processes	_	5
150 customers (terminal connection)	_	_
6 managers	2	4

Choosing Testing Hardware/Software

The hardware and software should be powerful and fast enough to emulate the required number of virtual users. Refer to the *HP LoadRunner Installation Guide* for specific hardware requirements.

To decide on the number of machines and correct configuration, consider the following:

➤ It is advisable to run HP LoadRunner Controller on a separate machine.

➤ Each GUI Vuser requires a separate Windows-based machine; several GUI Vusers can run on a single UNIX machine.

➤ Configuration of the test machine for GUI Vusers should be as similar as possible to the actual user's machine.

See the following tables to estimate the required hardware for each LoadRunner testing component. These requirements are for optimal performance.

Note: The results file requires a few MB of disk space for a long scenario run with many transactions. The load generators also require a few MB of disk space for temporary files if there is no NFS. See Chapter 11, "Configuring a Scenario" for more information about run-time file storage.

Examining Load Testing Objectives

Your test plan should be based on a clearly defined testing objective. This section presents an overview of common testing objectives:

➤ Measuring End-User Response Time

➤ Defining Optimal Hardware Configuration

➤ Checking Reliability

➤ Checking Hardware or Software Upgrades

➤ Evaluating New Products

➤ Identifying Bottlenecks

➤ Measuring System Capacity

Measuring End-User Response Time

Check how long it takes for the user to perform a business process and receive a response from the server. For example, suppose that you want to verify that while your system operates under normal load conditions, the end users receive responses to all requests within 20 seconds. The following graph presents a sample load vs. response time measurement for a banking application:
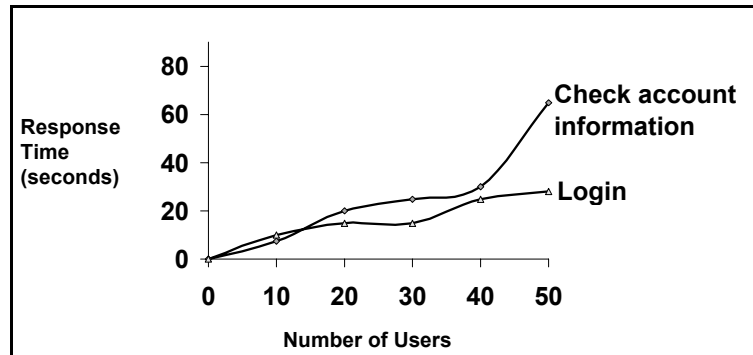
Defining Optimal Hardware Configuration

Check how various system configurations (memory, CPU speed, cache, adaptors, modems) affect performance. Once you understand the system architecture and have tested the application response time, you can measure the application response for different system configurations to determine which settings provide the desired performance levels.

For example, you could set up three different server configurations and run the same tests on each configuration to measure performance variations.

Checking Reliability

Determine the level of system stability under heavy or continuous work loads. You can use LoadRunner to create stress on the system: force the system to handle extended activity in a compressed time period to simulate the kind of activity a system would normally experience over a period of weeks or months.

Checking Hardware or Software Upgrades

Perform regression testing to compare a new release of hardware or software to an older release. You can check how an upgrade affects response time (benchmark) and reliability. Application regression testing does not check new features of an upgrade; rather it checks that the new release is as efficient and reliable as the older release.

Evaluating New Products

You can run tests to evaluate individual products and subsystems during the planning and design stage of a product's life cycle. For example, you can choose the hardware for the server machine or the database package based on evaluation tests.

Identifying Bottlenecks

You can run tests that identify bottlenecks on the system and determine which element is causing performance degradation, for example, file locking, resource contention, and network overload. Use LoadRunner in conjunction with the new network and machine monitoring tools to create load and measure performance at different points in the system. For more information, see Part VI, "Monitoring a Scenario."
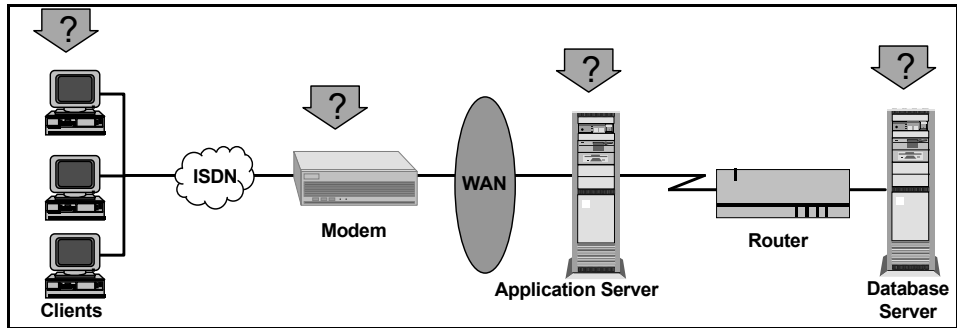
Measuring System Capacity

Measure system capacity, and determine how much excess capacity the system can handle without performance degradation. To check capacity, you can compare performance versus load on the existing system, and determine where significant response-time degradation begins to occur. This is often called the "knee" of the response time curve.
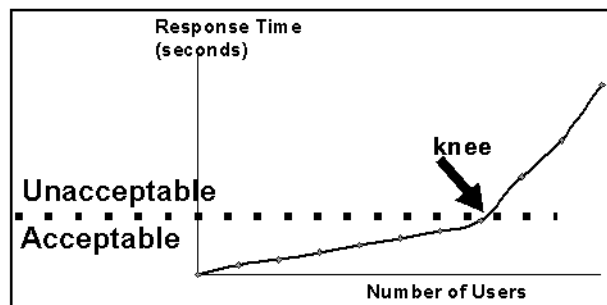
Once you determine the current capacity, you can decide if resources need to be increased to support additional users.

4

The Controller at a Glance

This chapter introduces the Controller window and explains how to perform basic scenario operations.

This chapter includes:

➤ Opening the Controller on page 48

➤ Introducing the Controller on page 51

➤ Managing Scenario Files on page 53

➤ Running a Scenario on page 56

Opening the Controller

Set up the LoadRunner environment according to the instructions in the *HP LoadRunner Installation Guide.*

To open the Controller:

You can open the Controller by choosing either of the following:

➤ **Start** > **Programs** > **LoadRunner** > **Applications** > **Controller.**

➤ **Start** > **Programs** > **LoadRunner** > **LoadRunner**. The HP LoadRunner launcher window opens. On the **Load Testing** tab, click **Run Load Tests**.

By default, the Controller opens with the New Scenario dialog box.

Note: To open the New Scenario dialog box on startup, ensure that the **Show at Startup** option in the New Scenario dialog box is selected. You can also select this option in the View menu (**View > Show New Scenario dialog**).

If the New Scenario dialog box does not open on startup, you can open it by seelcting **File > New** or by clicking the **New** button on the Controller toolbar.
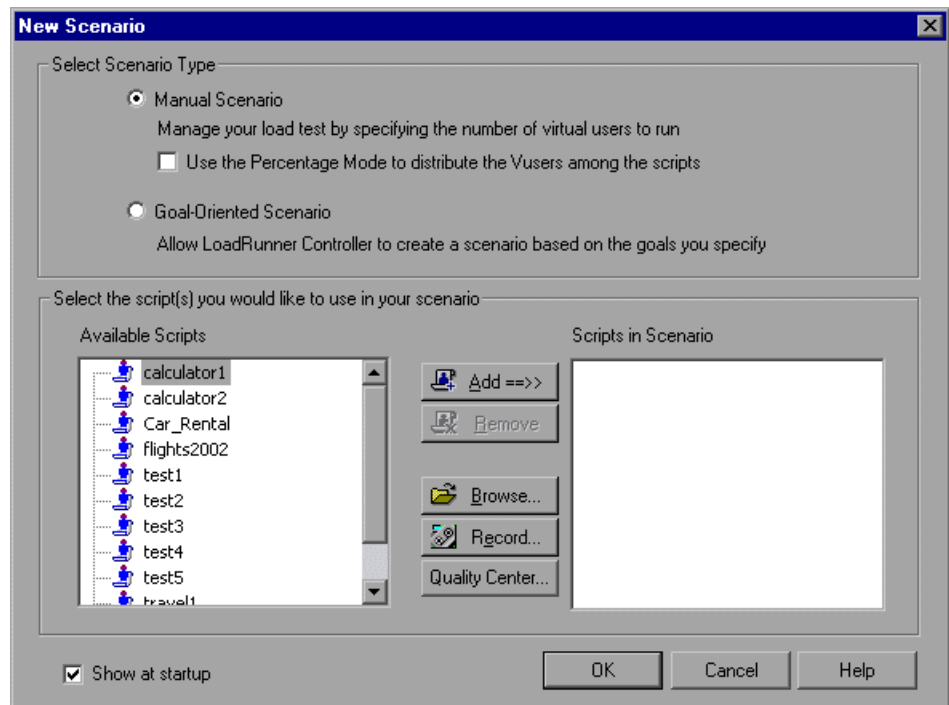
You can create one of the following types of scenario:

➤ **Manual Scenario.** You create the scenario yourself by defining the number of Vuser groups you want to run, and building a schedule for LoadRunner to run these groups. You can also create a manual scenario by defining the total number of Vusers to be used in the scenario, or assigning a percentage of the total number of Vusers to each script.

➤ **Goal-Oriented Scenario.** You define the goals you want your test to achieve, and LoadRunner automatically builds a scenario for you, based on these goals.

For instructions on creating a manual scenario, see Chapter 5, "Creating Manual Scenarios." For instructions on creating a manual scenario using the Percentage Mode, see Chapter 6, "Creating a Manual Scenario Using the Percentage Mode."

For instructions on creating a goal-oriented scenario, see Chapter 9, "Creating a Goal-Oriented Scenario."

To select scripts to use in your scenario:

1 Select a script from the Available Scripts list. By default, the list displays the 50 most recently used scripts.

Note: You can change the maximum number of scripts displayed in the Available Scripts list by modifying the following registry key: HKEY_CURRENT_USER\Software\Mercury Interactive\RecentScripts\ max_num_of_scripts

---

You can also click the **Browse** button to locate the script you want to use. To view the directory path of a script listed in the Available Scripts list, right-click the script and select **Show Paths**.

To select a script saved in the Quality Center database, click **Quality Center**. To record a new script using VuGen, click **Record**.

Note: To select a VB Vuser script, browse to locate the **.usr** file.

---

2 Click **Add** to copy the script you selected to the Scripts in Scenario list.

3 Click **Remove** to remove a script from the Scripts in Scenario list.

4 To bypass this dialog box the next time you create a new scenario, clear the **Show at startup** check box. You will be able to add scripts later on, while building your scenario.

5 Click **OK** to close the dialog box.

Introducing the Controller

The Controller window contains the following elements:

➤ **Title bar.** Displays the name of the scenario on which you are currently working.

➤ **Status Bar.** Displays tool tips for the Controller menu items, as well as the following, if they are enabled: Quality Center Connection, IP Spoofer, Auto Collate Results, and Auto Load Analysis.

Title bar

Design tab   Run tab   Status bar

The Controller window has the following views:

➤ **Design view.** This view displays a list of all the Vuser groups/scripts in a scenario, the load generators, and the number of Vusers assigned to each group/script. This view also displays basic information about the scenario schedule (manual scenario) or goal (goal-oriented scenario).

➤ **Run view.** Displays information on the running Vusers and Vuser groups, as well as online monitor graphs.

In addition, if you select **View > Show Output**, the Controller opens the Output window which displays error, warning, notification, debug, and batch messages generated during scenario execution.

Choosing Commands from the Toolbar

You can run many LoadRunner commands by clicking a button on the toolbar in the Controller. There are some variations in the buttons the toolbar displays, depending on whether you are in Design view or Run view, and depending on whether you are creating a manual scenario or a goal-oriented scenario.

In the Design view, the following buttons appear:

Toolbar Button	Enables You To
	Create a new scenario.
	Open an existing scenario.
	Save a scenario
	View load generators
	Open Vugen
	Open Analysis

In the Run view, the following additional buttons appear:

Toolbar Button	Enables You To
	Edit scenario goal (Goal-oriented scenarios only)
	Initialize Vusers
	Run Vusers
	Gradually stop running Vusers
	Stop running Vusers
	Analyze the results of the scenario

Managing Scenario Files

A scenario describes the events that occur during each load testing session. You create a scenario using the Design view of Controller.

After you create the scenario, LoadRunner saves the information in a scenario file (**.lrs**). You use the commands in the File menu to create, open, save, and close scenario files. Some of these commands are available from the toolbar.

Creating a New Scenario

The New command creates a completely new scenario. The New command clears all the information displayed in the Controller windows. To create a new scenario, select **File** > **New**, or click the **New** button on the Controller toolbar.

Opening an Existing Scenario

The Open command opens any existing scenario.

To open an existing scenario:

1 Select **File** > **Open**, or click the **Open** button. The Open Scenario dialog box opens.
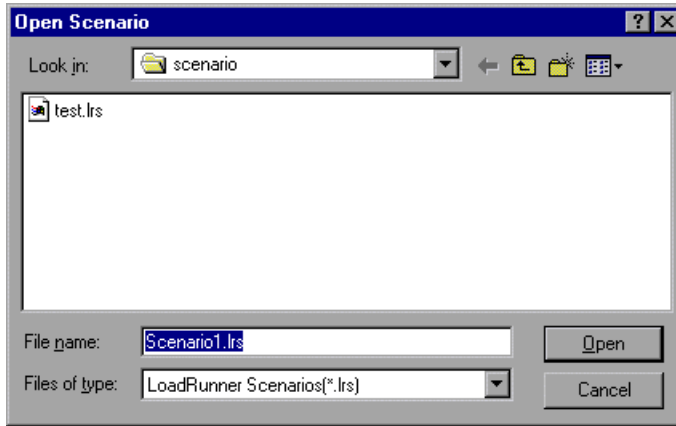
2 Click a file in the File Name list or type a file name in the **File Name** box.

3 Click **Open**. The File Open dialog box closes and the scenario appears in the Controller.

Saving a Scenario

The Save command saves the current scenario.

To save a scenario:

1 Select **File** > **Save**, or click the **Save** button. The Save Scenario dialog box opens the first time you save a scenario.
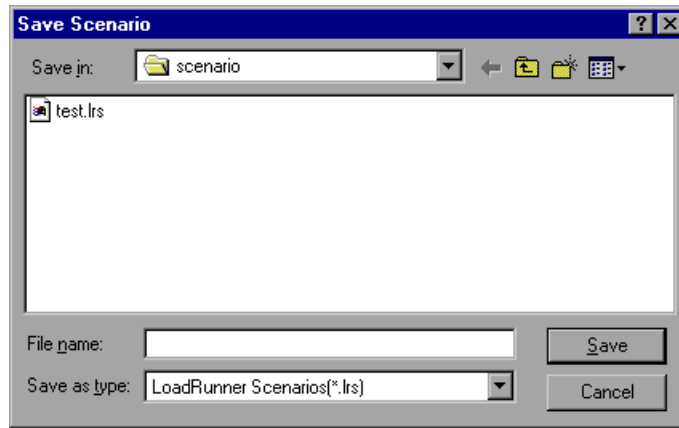
2 Type a scenario name in the File Name text box. By default, scenario files have the extension **.lrs**.

3 Click **Save**. The scenario is saved in the location you specified.

Closing a Scenario

Closing a scenario closes all the Controller windows. To close the scenario, select **File** > **Close**. If you made changes to the scenario, a Save Changes message appears. Click **Yes** to save the changes you made. All open windows and icons in the Controller close.

Running a Scenario

Once you have designed your scenario, you are ready to run it. You can control the Vusers and Vuser groups and monitor their performance online using the Run view of the Controller.

Scenario Groups pane

Scenario Status window

Online Monitors graph

Design tab

Run tab

During scenario execution, you use the Scenario Groups pane in the Run view to monitor the actions of all the Vusers and Vuser groups in the scenario. The Status field of each Vuser group displays the current state of each Vuser in the group.

You can also manipulate individual Vusers within the Vuser groups you have defined by selecting a group and clicking the **Vusers** button. The Vusers dialog box appears, with a list of the ID, Status, Script, Load Generator, and Elapsed Time (since the beginning of the scenario) for each of the Vusers in the group.

In addition, you can view a synopsis of the running scenario in the box in the upper-right corner of the Run view.

Scenario Status	Running	
Running Vusers	10	
Elapsed Time	00:00:38 (hh:mm:ss)	
Hits/Second	1.48 (last 60 sec)	
Passed Transactions	10	🔍
Failed Transactions	0	🔍
Errors	0	🔍

You can detach the Scenario Status window from the Run view, thereby enlarging the Scenario Groups pane.

While the scenario runs, the Vusers and load generators send error, notification, warning, debug, and batch messages to the Controller. You can view these messages in the Output window (**View** > **Show Output**).
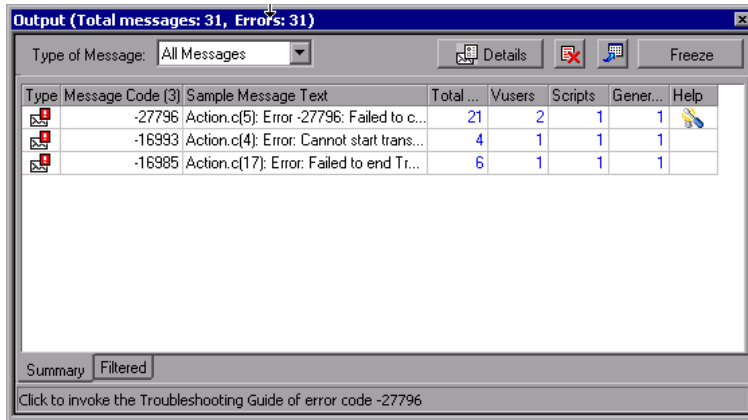
For more information on the Output window, see "Viewing the Output Window" on page 289.

You use the online monitors and online monitor graphs to monitor Vuser status, transactions, system resources, database server resources, Web server resources, Web application server resources, network delay, streaming media resources, firewall server resources, ERP/CRM server resources, Java performance, J2EE & .NET diagnostics, application deployment resources, middleware performance, application component resources, and infrastructure resources while running a scenario. For more information on online monitors, see Chapter 24, "Online Monitoring."

Part II

Designing a Scenario

5

Creating Manual Scenarios

You can build a manual scenario by creating groups and specifying the scripts, the load generators, and the number of Vusers included in each group. You can also build a manual scenario using the Percentage Mode, which allows you to define the total number of Vusers to be used in the scenario, and assign load generators and a percentage of the total number of Vusers to each script.

This chapter describes how to create a manual scenario using the Vuser Group Mode. For information on creating a manual scenario using the Percentage Mode, see Chapter 6, "Creating a Manual Scenario Using the Percentage Mode."

This chapter includes:

About Creating a Scenario

To test your system with LoadRunner, you must create a scenario—a file with information about the test session. The scenario is the means by which you emulate a real-life user. The scenario contains information about how to emulate real users: the groups of virtual users (Vusers), the test scripts the Vusers will run, and the load generators upon which to run the scripts.

If you chose to create a regular manual scenario, each script you selected in the New Scenario dialog box is assigned to a Vuser group. To each Vuser group you then assign a number of virtual users. You can instruct all Vusers in a group to run the same script on the same load generator, or you can assign different scripts and load generators to the various Vusers in a group.

Once you create your Vuser groups, you select or build a schedule for your scenario. See Chapter 7, "Scheduling Scenarios" for more information on creating a scenario schedule.

Understanding the New Scenario Dialog Box

The New Scenario dialog box enables you to select scripts for a new scenario.
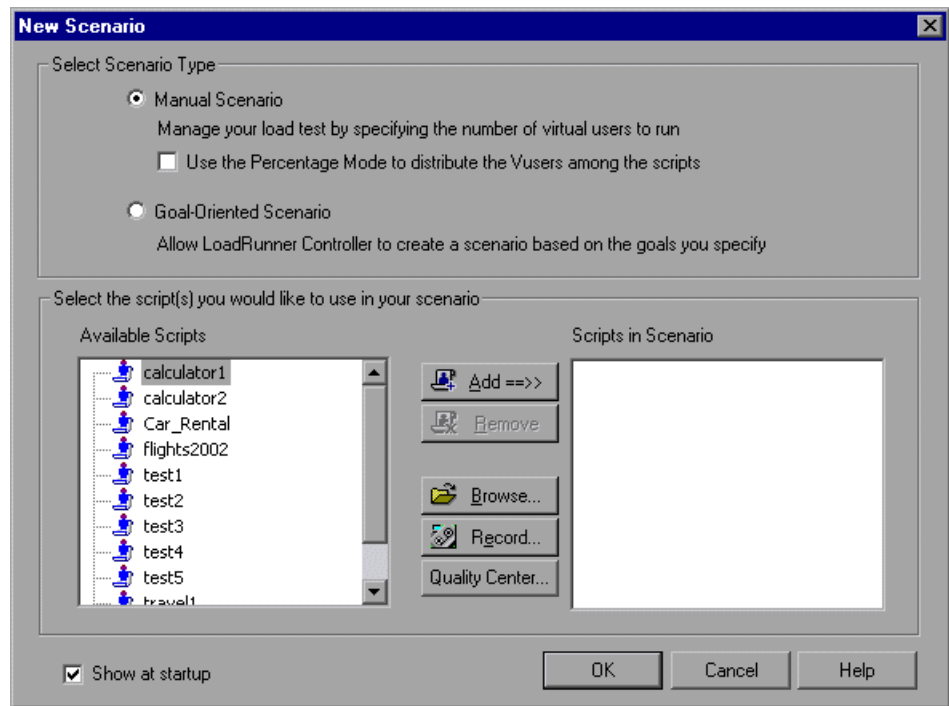
Selecting the Scenario Type

Select one of the following scenario options:

➤ **Manual Scenario.** Select this method if you want to build a manual scenario. You build a manual scenario by creating groups and specifying the script, the load generator, and the number of Vusers included in each group.

➤ **Use the Percentage Mode to distribute the Vusers among the scripts.** Select this option if you want to build a manual scenario by specifying a number of Vusers to be distributed among the selected Vuser scripts.

➤ **Goal Oriented Scenario.** Select this method to have LoadRunner build a scenario for you. In a goal-oriented scenario, you define the goals you want your test to achieve, and LoadRunner automatically builds a scenario for you, based on these goals.

Choosing a Script

Select a script from the Available Scripts list. Scripts that have been selected appear in the Scripts in Scenario pane.

➤ **Available Scripts.** Displays, by default, a list of the fifty most recently used scripts.

Note: You can change the maximum number of scripts displayed in the Available Scripts list by modifying the following registry key: **HKEY_CURRENT_USER\Software\Mercury Interactive\RecentScripts\ max_num_of_scripts**

➤ **Add.** Adds a script to the scenario.

➤ **Remove.** Removes a script from the scenario.

➤ **Browse.** Select scripts from a different directory. To select a VB Vuser script, browse to locate the **.usr** file.

➤ **Record.** Opens the Virtual User Generator so that you can begin recording a script. For more information on recording scripts, see the *HP Virtual User Generator User's Guide*.

➤ **Quality Center.** Opens the Connection to Quality Center dialog box, so that you can open a connection to a Quality Center project.

➤ **Scripts in Scenario.** Displays the scripts to be used in the scenario.

➤ **Show at startup.** When selected, LoadRunner displays the New Scenario dialog box each time you open the Controller.

Creating Vuser Groups

A scenario consists of groups of Vusers that emulate human users interacting with your application. When you run a scenario, the Vusers generate load on the server, and LoadRunner monitors the server and transaction performance.

Vuser groups are used to organize the Vusers in a scenario into manageable groups. You create Vuser groups that contain Vusers with shared or similar characteristics. For example, you can create a Vuser group for all Vusers that run the same Vuser script.

Understanding the Design Tab - Manual Scenario Mode

When you create a manual scenario, the Controller's Design tab displays the Scenario Groups, Service Level Agreement, and Scenario Schedule panes.

The **Scenario Groups** pane lists all enabled and disabled Vuser Groups, their paths, the number of Vusers assigned to each group, and the load generators.

The **Scenario Schedule** pane displays information regarding the schedule profile: its name, the schedule mode, the scenario run mode, the total number of Vusers to be used in the scenario, and the load behavior. The Interactive Schedule graph displays a graph of the scenario schedule you defined. For more information on creating a scenario schedule, see Chapter 7, "Scheduling Scenarios."

The **Service Level Agreement** pane lists all the service level agreements (SLAs) defined for the scenario. For more information in SLAs, see Chapter 8, "Defining Service Level Agreements."

You can perform the following actions on a Vuser group or scenario:

➤ Define the group name, Vuser quantity, load generators, and scripts for the Vuser group

➤ Add one or more load generators to the Vuser group and configure the load generators

➤ Add and configure one or more scripts to the Vuser group

➤ Enable or disable a Vuser group for the scenario

➤ Remove a Vuser group from the scenario

➤ Schedule the Vuser group/scenario

➤ Define service level agreements for the scenario

➤ Run the scenario

➤ Stop the scenario

➤ Reset the scenario

➤ Configure scenario result settings

Adding a Vuser Group

Vuser groups can be created and added to a scenario using the Add Group
dialog box.

To create Vuser Groups:

1 On the Scenario Groups pane toolbar, click the **Add Group** button. The Add
Group dialog box opens:
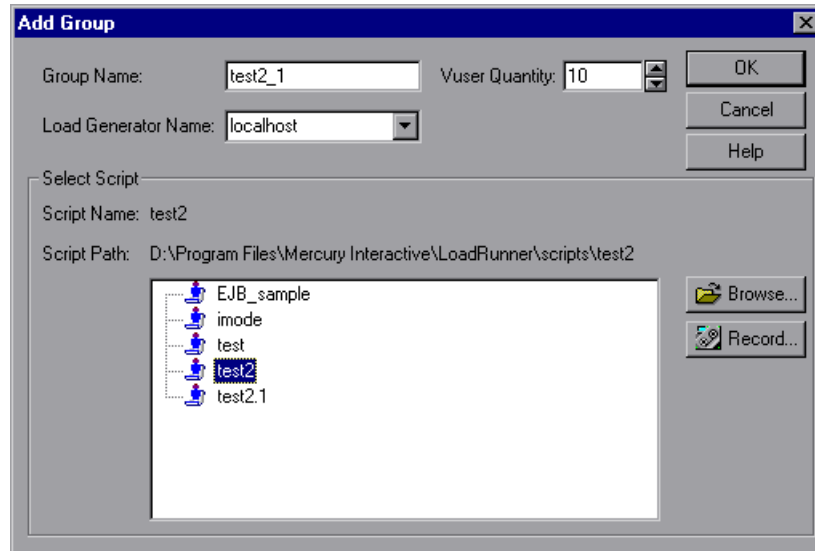
2 In the Group Name box, enter a name for the Vuser group.

3 From the Vuser Quantity box, select the number of Vusers that you want to
create in the group.

4 Select a load generator from the Load Generator Name list. To use a load generator that does not appear, select **Add** from the Load Generator Name list. The Add New Load Generator dialog box opens:
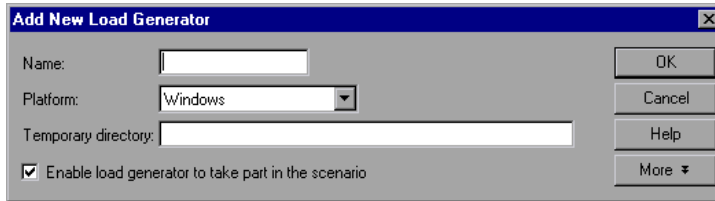
Type the name of the load generator in the Name box. In the Platform box, select the type of platform on which the load generator is running.

By default, LoadRunner stores temporary files on the load generator during scenario execution, in a temporary directory specified by the load generator's TEMP or TMP environment variables. To override this default for a specific load generator, type a location in the Temporary Directory box.

To allow the load generator to take part in the scenario, check **Enable load generator to take part in the scenario**.

Click **More** to expand the dialog box and show the Add Load Generator tabs. For information on configuring settings for each load generator, see "Configuring Load Generator Settings" on page 88.

Click **OK** to close the Add New Load Generator dialog box.

5 Select a script from the script list.

To use a script that does not appear, click **Browse**. Browse to select the path and file name of the new script.

Note: If a script uses Unique file parameterization, running more than one Vuser group with that script in the same scenario may cause unexpected scenario results. For more information about Unique file parameterization, see the *HP Virtual User Generator User's Guide*.

---

6 Click **OK** to close the Add Group dialog box. The new group's properties appear in the Scenario Groups pane.

Understanding the Add Group Dialog Box

Use the Add Group dialog box to insert a new group into the scenario.

➤ **Group Name.** Enter the name of the new group you want to add. The name is limited to a maximum of 55 characters.

➤ **Vuser Quantity.** Select the number of Vusers you want to add to the group.

➤ **Load Generator Name.** Select the name of the load generator for the new group. Choose a previously existing load generator from the list or create a new load generator by choosing **Add**. The Add Load Generator dialog box opens.

➤ **Select Script.** Displays the available scripts in the current directory. The list contains all scripts that you previously added to the scenario.

   ➤ **Script Name.** Select the script you want the Vuser group you are creating to use. The script appears in the Script Name column.

   ➤ **Script Path.** Displays the script directory's path.

   ➤ **Browse.** Select the path and file name of a script from a different directory. To use a VB Vuser script, select the **.usr** file.

   Note: When you specify the location of a script, you can specify a location that is relative to the current scenario directory. For details, see "Using Relative Paths for Scripts" on page 112.

   ➤ **Record.** Opens the Virtual User Generator so that you can begin recording a script. For more information on recording scripts, see the *HP Virtual User Generator User's Guide*.

   Note: While a scenario is running, you can add Vuser groups to the scenario and enable them. However, if you add a Vuser group after all the Vusers in the scenario have started running, the new group will not run in the scenario.

Deleting a Vuser Group

To delete a Vuser group, on the Scenario Groups pane toolbar, click the **Remove Group** button, or right-click the Vuser group and select **Remove Group**.

Disabling a Vuser Group

By default, all the Vuser groups displayed in the Scenario Groups pane are enabled to run in the scenario. To disable a Vuser group, clear the check box to the left of the Vuser group name. The color of the group changes to gray, indicating that the group will not take part in the scenario. To enable the Vuser group, select the check box again.

Modifying a Vuser Group

You can modify the script, and load generator for a Vuser group directly from the Scenario Groups pane of the Controller, or by using the Group Information dialog box.

To modify a Vuser Group directly from the Scenario Groups pane:

1 Select the group name, script path, or load generator you want to modify, and edit its properties.

2 To modify a Vuser group script's run-time settings, on the Scenario Groups pane toolbar, click the **Run-Time Settings** button. For more information about run-time settings, see "Configuring Scripts" on page 107.

3 To edit a Vuser group's script, on the Scenario Groups pane toolbar, click the **View Script** button. LoadRunner's script generation tool, VuGen, opens. For more information on editing scripts, see the *HP Virtual User Generator User's Guide*.

To modify a Vuser Group using the Group Information dialog box:

 1 On the Scenario Groups pane toolbar, click the **Details** button, or right-click the property you want to modify and select **Details**. The Group Information dialog box opens.
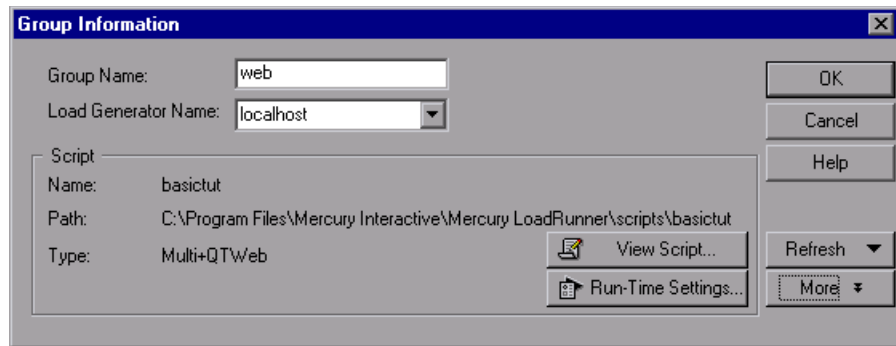
 2 In the **Group Name** box, enter the Vuser group name.

 3 Select a load generator from the **Load Generator Name** list.

To use a load generator that does not appear, select **<Add...>** from the **Load Generator Name** list and add a new load generator using the Add Load Generator dialog box.

 4 To modify the run-time settings you specified while recording a script using VuGen, click **Run-Time Settings**. For more information about run-time settings, see "Configuring Scripts" on page 107.

 5 To edit a Vuser group's script, click **View Script**. LoadRunner's script generation tool, VuGen, opens. For more information on editing scripts, see "Configuring Scripts" on page 107.

 6 Click **OK** to close the Group Information dialog box.

Understanding the Group Information Dialog Box

Use the Group Information dialog box to display details about a Vuser group, and modify the settings of a group.

➤ **Group Name.** Displays the current group name. To modify the name, type a new name in the Group Name box.

➤ **Load Generator Name.** Displays the name of the selected Vuser's load generator. To specify a different load generator, select one from the Load Generator Name list. Select **Add** from the Load Generator Name list to specify a new load generator.

➤ **Vuser Quantity.** Displays the number of Vusers in the group.

➤ **Script.** Displays details of the selected script.

   ➤ **Name.** Displays the name of the script.

   ➤ **Path.** Displays the script directory's path.

   ➤ **Type.** Displays the type of script.

   ➤ **View Script.** Opens the Virtual User Generator, so that you can edit the script. For more information on editing scripts, see the *HP Virtual User Generator User's Guide*.

   ➤ **Run-Time Settings.** Opens the Run-Time Settings dialog box, enabling you to edit the script run-time settings you previously set using VuGen. If you did not set run-time settings for a script in VuGen, the default VuGen settings are displayed for all but the Log and Think Time tabs, which display the default Controller settings. For information on the run-time settings, see the *HP Virtual User Generator User's Guide*.

➤ **Refresh.** Click this button and select **Script** to update the script details in the scenario, if you make any changes to a script while the Controller is running. Select **Run-Time Settings** to restore the initial run-time settings, if you modify the run-time settings from the Controller.

➤ **More/Less.** Reveals/hides the following:

   ➤ **Command Line.** Type any command line options to use when running the script—for example: -x value -y value. For information about passing command line argument values to a script, see the *HP Virtual User Generator User's Guide*.

   ➤ **Rendezvous.** Displays the rendezvous points defined for the selected script.

   ➤ **Vusers.** Displays all Vusers associated with the selected script.

   ➤ **Files.** Displays all files used by the script. To exclude a file from the list, clear the check box adjacent to it.

➤ **Add.** Click this button and select **Add File** or **Add Directory**. Browse to the file or directory that you want to add to this list and click **Open**. The selected files/directories are added to the files list.

Note: To run Visual C++ Vusers on a remote load generator, you must add the **.dll** of the Vuser to the files used by script list.

Sorting Vuser Groups in the Scenario Groups Pane

Once you have created your Vuser groups, you can sort them by group name, script name, quantity of Vusers assigned to the group, or load generator name.

To sort Vuser groups:

➤ Select the column by which you want to sort the groups. Click the column header.

➤ Alternatively, you can right-click anywhere within the column you want to sort, and select **Sort Groups**. Choose to sort by name, path, quantity, or load generator.

➤ To instruct the Controller to automatically sort a new Vuser group entry, right-click the entry and select **Auto Sort**.

Configuring Vusers in a Vuser Group

You can define properties for individual Vusers within the Vuser groups you have defined, using the Vusers dialog box. To each Vuser you can assign a different script and/or load generator.

To define properties for individual Vusers:

1 Select the Vuser group whose Vusers you want to modify, and, on the Scenario Groups pane toolbar, click the **Virtual Users** button. The Vusers dialog box opens.
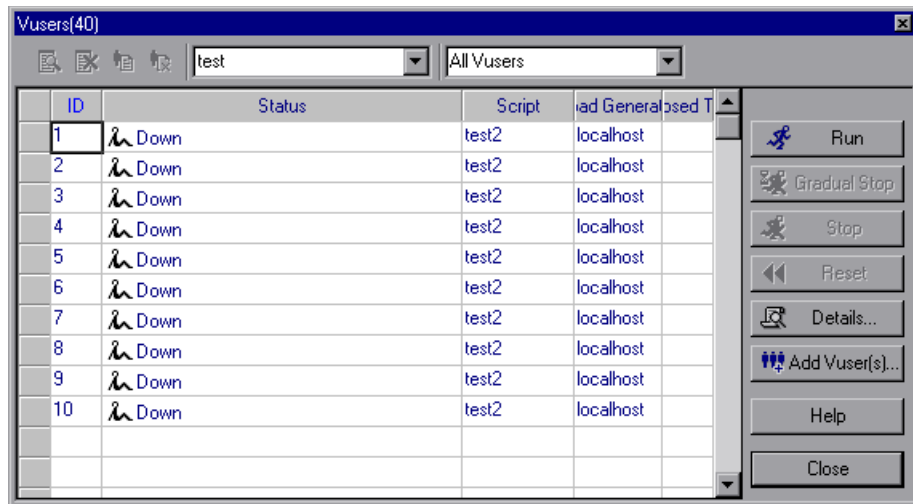
2 To change the script for an individual Vuser, select a different script in the Script column. Alternatively, you can click the **Details** button, and select a different script from the script list in the Vuser Information dialog box.

3 To change the load generator on which a Vuser runs, select a different load generator in the **Load Generator** column. Alternatively, you can click the **Details** button, and select a different load generator from the **Load Generator Name** list in the Vuser Information dialog box.

To use a load generator that does not appear, in the **Load Generator Name** list select **<Add..>** and add a new load generator using the Add Load Generator dialog box.

Understanding the Vusers Dialog Box

The Vusers dialog box displays the status of the Vusers in the group.

Choose the scenario group from the list at the top of the dialog box.

➤ **Show the Selected Vusers.** Opens a Run-Time Viewer for each selected Vuser.

➤ **Hide the Selected Vusers.** Closes the Run-Time Viewers that were opened.

➤ **Open Vuser Log.** Displays a log containing run-time information about the Vuser that is refreshed, by default, every 1000 milliseconds.

➤ **Close Vuser Log.** Closes the Vuser log.

➤ **ID.** Displays the Vuser's ID number.

➤ **Status.** Displays the Vuser's status. The possible statuses are:

Status	Description
Down	The Vuser is down.
Pending	The Vuser is ready to be initialized and is waiting for an available load generator, or is transferring files to the load generator. The Vuser will run when the conditions set in its scheduling attributes are met.
Initializing	The Vuser is being initialized on the remote machine.
Ready	The Vuser already performed the init section of the script and is ready to run.
Running	The Vuser is running. The Vuser script is being executed on a load generator.
Rendezvous	The Vuser has arrived at the rendezvous and is waiting to be released by LoadRunner.
Done.Passed	The Vuser has finished running. The script passed.
Done.Failed	The Vuser has finished running. The script failed.
Error	A problem occurred with the Vuser. Check the Status field on the Vuser dialog box or the output window for a complete explanation of the error.

Status	Description
Gradual Exiting	The Vuser is completing the iteration or action it is running (as defined in **Tools** > **Options** > **Run-Time Settings**) before exiting.
Exiting	The Vuser has finished running or has been stopped, and is now exiting.
Stopped	The Vuser stopped when the Stop command was invoked.

➤ **Script.** Displays the script run by the Vuser.

➤ **Load Generator.** Displays the load generator on which the Vuser is running.

➤ **Elapsed Time.** Displays the amount of time that has elapsed in the scenario since the Vuser began running.

➤ **Run.** Instructs the Controller to begin running the Vuser.

➤ **Gradual Stop.** Instructs the Controller to complete the current iteration or action before stopping the Vuser. This option is only available when the Vuser is in the RUN state, if you selected the **Wait for the current iteration to end before exiting** or **Wait for the current action to end before exiting** options in the Run-Time Settings tab of the Options dialog box.

➤ **Stop.** Instructs the Controller to stop running the Vuser immediately.

➤ **Reset.** Resets the status of the Vuser to **Down**.

➤ **Details.** Opens the Vuser Information dialog box.

➤ **Add Vusers.** Opens the Add Vusers dialog box so that you can add one or more Vusers.

The following additional right-click options are available:

➤ **Reset IDs.** Resets the IDs of the Vusers in the group.

➤ **Run-Time Settings.** Opens the Run-Time Settings dialog box, enabling you to edit the script run-time settings you previously set using VuGen. If you did not set run-time settings for a script in VuGen, the default VuGen settings are displayed for all but the Log and Think Time tabs, which display the default Controller settings. For information on the run-time settings, see the *WinRunner User's Guide*.

Changing the run-time settings for one Vuser changes the run-time settings for all of the Vusers in the group.

➤ **View Script.** Opens the Virtual User Generator so that you can edit the script. For more information on editing scripts, see the *HP Virtual User Generator User's Guide*.

➤ **Initialize Vusers.** Distributes the Vuser to its designated load generator so that it is ready to execute its script. If a Vuser fails to initialize, its status changes to **Error**.

➤ **Pause Vusers.** Temporarily pauses the Vuser's execution of its script.

Note: Pausing a Vuser group will affect its transaction response time.

➤ **Show Vuser.** Opens the Run-Time Viewer and displays the Vuser executing its script.

➤ **Hide Vuser.** Closes the Run-Time Viewer showing the Vuser executing its assigned script.

➤ **Show Vuser Log.** Displays a log containing run-time information about the Vuser that is refreshed, by default, every 1000 milliseconds.

➤ **Hide Vuser Log.** Closes the Vuser script log.

➤ **Filter Vusers.** Filters the Vusers displayed in the Vusers dialog box by status. You can also select the filter option you want to use from the filter selector at the top of the Vusers dialog box.

➤ **Sort Vusers.** Sorts the Vusers in the group by ID, Status, Script, Load Generator, or Elapsed Time.

Understanding the Vuser Information Dialog Box

The Vuser Information dialog box displays details about a specific Vuser in a group, and lets you modify the load generator and script settings for the Vuser.

➤ **Group Name.** Displays the name of the group to which the selected Vuser belongs.

➤ **Vuser Name.** Displays the name of the selected Vuser.

➤ **Load Generator Name.** Displays the name of the selected Vuser's load generator. To specify a different load generator, select one from the Load Generator Name list. Select **Add** to specify a new load generator.

➤ **Select Script.** Displays the available scripts in the current directory.

➤ **Script Name.** Select the script you want the Vuser you selected to use. The script appears in the Script Name column.

➤ **Script Path.** Displays the script directory's path.

➤ **Browse.** Select a script from a different directory. To select a VB Vuser script, browse to locate the **.usr** file.

➤ **Record.** Opens VuGen so that you can begin recording a script. For more information on recording scripts, see the *HP Virtual User Generator User's Guide*.

➤ **Run-Time Settings.** Opens the Run-Time Settings dialog box, enabling you to edit the script run-time settings you previously set using VuGen. If you did not set run-time settings for a script in VuGen, the default VuGen settings are displayed for all but the Log and Think Time tabs, which display the default Controller settings. For information on the run-time settings, see the *HP Virtual User Generator User's Guide*.

➤ **Parameter List**. Opens the Parameter List in VuGen. For information about the Parameter List, see the *HP Virtual User Generator User's Guide*.

Adding Vusers to a Vuser Group

You add Vusers to a Vuser group and define their properties using the Add Vusers dialog box.

Note: You can activate additional Vusers while the scenario is running, using the Run/Stop Vusers dialog box. For more information, see "Manually Adding Vusers to a Running Scenario" on page 278.

To add Vusers to a Vuser group:

 1 In the Vusers dialog box, click the **Add Vusers** button. The Add Vusers dialog box opens.
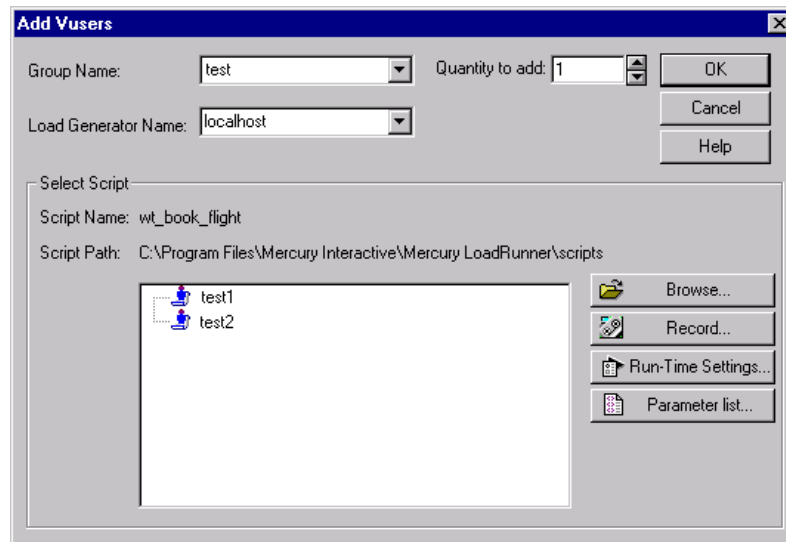
 2 From the Group Name box, select the name of the Vuser group.

 3 From the Quantity to add box, select the number of Vusers that you want to add to the group.

4 Select a load generator from the Load Generator Name list.

To use a load generator that does not appear, select **Add** from the Load Generator Name list and add a new load generator using the Add Load Generator dialog box.

5 Select a script from the script list.

To use a script that does not appear, click the **Browse** button. Browse to select the path and file name of the new script.

6 Click **OK** to close the Add Vusers dialog box. The new Vuser's properties appear in the Vusers dialog box.

Understanding the Add Vusers Dialog Box

Adds a new Vuser to the Vuser group.

➤ **Group Name.** Enter the name of the group to which you want to add a Vuser.

➤ **Load Generator Name.** Select the name of the load generator for the new Vuser. Choose a previously existing load generator from the list or create a new load generator by choosing **Add**. The Add Load Generator dialog box opens.

➤ **Quantity to add.** Select the number of Vusers you want to add to the group.

➤ **Select Script.** Displays the available scripts in the current directory.

  ➤ **Script Name.** Select the script you want the Vuser you are creating to use. The script appears in the Script Name column.

  ➤ **Script Path.** Displays the script directory's path.

  ➤ **Browse.** Select a script from a different directory. To use a VB Vuser script, select the **.usr** file.

  Note: When you specify the location of a script, you can specify a location that is relative to the current scenario directory. For details, see "Using Relative Paths for Scripts" on page 112.

  ➤ **Record.** Opens VuGen so that you can begin recording a script. For more information on recording scripts, see the *HP Virtual User Generator User's Guide*.

➤ **Run-Time Settings.** Opens the Run-Time Settings dialog box, enabling you to edit the script run-time settings you previously set using VuGen. If you did not set run-time settings for a script in VuGen, the default VuGen settings are displayed for all but the Log and Think Time tabs, which display the default Controller settings. For information on the run-time settings, see the *HP Virtual User Generator User's Guide*.

Note: Modifying the run-time settings for the new Vuser will modify the run-time settings for all the Vusers in the group. For more information about run-time settings, see "Configuring Scripts" on page 107.

➤ **Parameter List**. Opens the Parameter List in VuGen. For information about the Parameter List, see the *HP Virtual User Generator User's Guide*.

Configuring Vuser Run-Time Settings

You set the script's run-time settings to customize the way the Controller executes a Vuser script.

To view the run-time settings in the Controller:

In the Controller's Design tab, highlight a group or groups in the Scenario Groups pane, and click **Run-Time Settings**.

The Run-Time Settings dialog box displays the settings you previously set using VuGen. If you did not set run-time settings for a script in VuGen, the default VuGen settings are displayed for all but the Log and Think Time tabs, which display the default Controller settings.

Note: Several protocols, such as Web and Java, have specific settings.

For information on each specific run-time setting, see the *HP Virtual User Generator User's Guide*.

Modifying the run-time settings for the new Vuser will modify the run-time settings for all the Vusers in the group. If a group contains more than one Vuser type, then you can modify the shared run-time settings, as described in "Modifying Run-Time Settings for Multiple Scripts" below.

Note: If you modify the run-time settings from the Controller, LoadRunner runs the script using the modified settings. To restore the initial settings, click the **Refresh** button and select **Run-Time Settings**.

Modifying Run-Time Settings for Multiple Scripts

When you choose to modify script run-time settings, and you have selected multiple scripts, or a group with multiple scripts, the Controller displays the option of modifying the shared run-time settings:
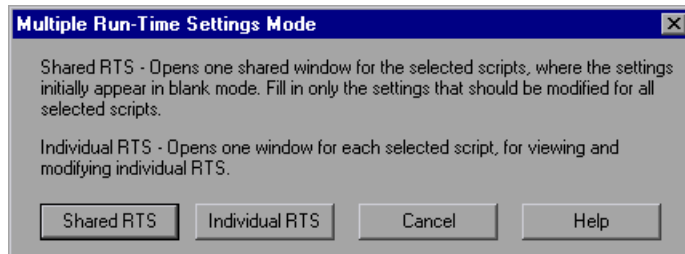
Note: If one of the selected scripts does not support shared run-time settings, then you will only have the option of modifying each script's individual run-time settings. Shared RTS mode will be disabled for GUI or Astra LoadTest Vusers.

Select a method for modifying run-time settings of multiple scripts:

➤ **Shared RTS.** Opens one window containing all of the run-time settings in blank mode. In this mode, you set only the options that you would like to modify for all selected scripts. All other run-time settings remain unchanged.

➤ **Individual RTS.** Opens a separate window for each selected script. In this mode, you modify each script's settings individually.

Modifying Shared Run-Time Settings

Any settings that you change in shared mode will be applied to all selected scripts. Any other settings remain unchanged. For example, if a dialog has checkboxes, they appear in disabled mode, meaning that they are neither selected nor cleared. If you select or clear a check box, then the change will apply to all selected scripts.

Some run-time settings cannot be modified in shared mode. These settings will not appear. To modify them, open the run-time settings for each individual script.

All Run-Time Setting buttons will be disabled, for example the **Change** and **Advanced** buttons in the Browser Emulation node.

The following nodes will not appear in shared mode:

➤ the **Java Environment Settings:Classpath** node

➤ the **Internet Protocol:ContentCheck** node

➤ the **Run Logic** node - for protocols which support the **Run Logic** node, the **Iterations** box will appear in the **Pacing** node

➤ the nodes with tables in the format **Property, Value** for the protocols: Citrix ICA, Oracle NCA, and WAP, for example, the **Oracle NCA:Client Emulation** node

Configuring Load Generators

You can set a load generator's attributes while adding it to the load generator list, or modify the attributes of an existing load generator at any time, using the Load Generators dialog box.

To configure global settings for all load generators participating in the scenario, use LoadRunner's Options dialog box. For more information, see Chapter 11, "Configuring a Scenario." To set properties specific for each load generator, use the Load Generators dialog box as described below.

You can also indicate which load generators will run Vusers in the scenario. For example, if a load generator is unavailable for a particular scenario run, you can exclude it temporarily instead of removing it entirely from your list of load generators.

You select which load generators will take part in the scenario by using the Enable and Disable commands. Disabling a load generator temporarily removes it from the list. Enabling a load generator reinstates it. Disabling load generators is particularly useful if you want to isolate a specific machine to test its performance.

To configure a load generator:

 1 Click the **Load Generators** button, or select **Scenario** > **Load Generators**. The Load Generators dialog box opens. The **Name** of the load generator, its **Status**, **Platform**, and **Details** are displayed.
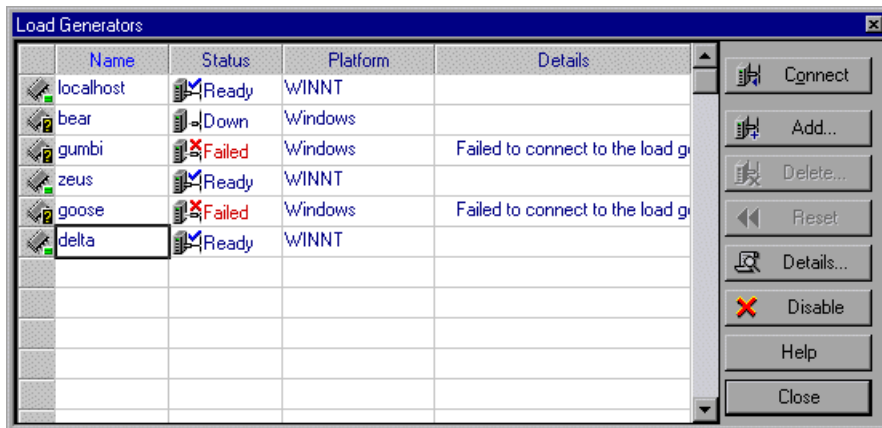
2 Click **Connect** to change the Status of the load generator from **Down** to **Ready**. When the load generator is connected, the button automatically changes to **Disconnect**. To change the Status of the load generator from **Ready** to **Down**, click **Disconnect**.

3 To disable a load generator, select the load generator and click **Disable**. The load generator name changes from blue to gray, and the load generator is disabled. To enable a load generator, select the load generator and click **Enable**. The load generator name changes from gray to blue, and the load generator is enabled.

4 To view details of a load generator, select the load generator and click **Details**. The Load Generator Information dialog box opens with information about the load generator you selected.

5 To remove a load generator, select it from the list and click **Delete**.

Understanding the Load Generators Dialog Box

The Load Generators dialog box displays information about the load generators connected to the scenario.

➤ **Name.** Lists the name of the load generator.

➤ **Status.** Displays the status of the load generator. The following table describes the possible statuses of the load generator.

Status	Description
Ready	The load generator is connected
Connecting	The load generator is in the process of connecting
Active	The load generator is running Vusers
Down	The load generator is not connected
Failed	A connection with the load generator could not be established

➤ **Platform.** Displays the type of platform on which the load generator is running.

➤ **Details.** In the event that the connection fails, displays details about why it failed.

➤ **Connect.** Instructs the Controller to connect the load generator for the scenario. When the load generator is connected, the button automatically changes to **Disconnect**.

➤ **Add.** Opens the Add Load Generator dialog box.

➤ **Delete.** Deletes the load generator. The load generator can be deleted only when it is disconnected.

➤ **Reset.** Attempts to reset a failed connection.

➤ **Details.** Opens the Load Generator Information dialog box.

➤ **Disable/Enable.** Instructs the Controller to disable or enable the load generator. When a load generator is disabled, its Name, Status, Platform, and Details appear in gray.

Note: The Controller monitors a Windows load generator's CPU usage and automatically stops loading Vusers on a load generator when it becomes overloaded. You can monitor the status of a machine's CPU usage using the icons in the Load Generator dialog box. When the CPU usage of a load generator becomes problematic, the icon to the left of the load generator name contains a yellow bar. When the machine becomes overloaded, the icon contains a red bar.

Adding a Load Generator

You can add load generators to your scenario, or modify information for existing load generators.

To add a load generator, or modify load generator information:

1 Click **Add** in the Load Generators dialog box. The Add New Load Generator dialog box opens.
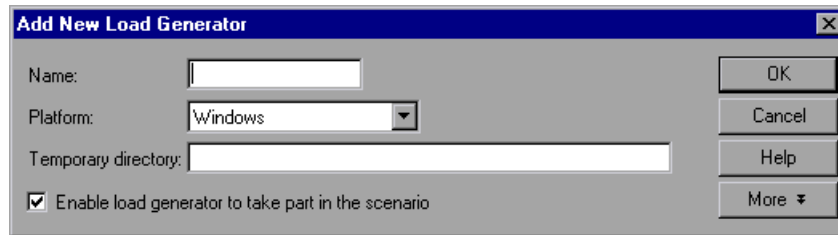
2 In the **Name** box, enter the name of the load generator.

3 In the **Platform** box, select the type of platform on which the load generator is running.

4 In the **Temporary Directory** box, type a location on the load generator, where the Controller can store temporary files, or leave empty to accept the default location. By default, LoadRunner stores temporary files on the load generator during scenario execution, in a temporary directory specified by the load generator's TEMP or TMP environment variables.

5 To allow the load generator to take part in the scenario, check **Enable load generator to take part in the scenario**.

6 Click **More** to expand the dialog box and show the Add New Load Generator tabs. For information on configuring these settings, see "Configuring Load Generator Settings" on page 88.

7 Click **OK** to close the Add New Load Generator dialog box. The load generator name you entered appears in the Load Generators list; its status is set to **Down**.

Understanding the Add New Load Generator Dialog Box

You can add load generators to your scenario using the Add New Load Generator dialog box.

➤ **Name.** Type the name of the load generator you want to add in the Name box.

➤ **Platform.** Select the type of platform on which the load generator is running.

➤ **Temporary Directory.** Type a location, on the load generator, where the Controller can store temporary files. By default, LoadRunner stores temporary files on the load generator during scenario execution, in a temporary directory specified by the load generator's TEMP or TMP environment variables.

➤ **Enable load generator to take part in the scenario.** Select to include the load generator in the scenario.

➤ **More/Less**. Reveals/hides Status, Run-Time File Storage, UNIX Environment, Run-Time Quota, Vuser Status, Vuser Limits, Terminal Services, Security, and Connection Log (Expert Mode) tabs.

Configuring Load Generator Settings

You can configure additional settings for individual load generators, using the tabs in the Add New Load Generator or Load Generator Information dialog boxes. You can configure **Status**, **Run-Time File Storage**, **UNIX Environment**, **Run-Time Quota**, **Vuser Limits**, **Vuser Status**, **Terminal Services**, and **Security** settings. You can also configure the Connection Log settings while working in Expert mode. For information on the Connection Log tab, see Appendix C, "Working in Expert Mode."

You can configure global settings for all load generators participating in the scenario, using the Options dialog box. For more information, see Chapter 11, "Configuring a Scenario."

To configure load generator settings:

1 Click the **Load Generators** button , or select **Scenario** > **Load Generators** to open the Load Generators dialog box.

2 Select a load generator in the list and click the **Details** button. The Load Generator Information dialog box opens.

You can also configure the settings from the Add New Load Generator dialog box. Click **More** to display the settings tabs.

3 Select a tab, and configure the load generator settings.

The settings apply to the load generator specified in the **Name** box. To configure a load generator other than the one specified, enter the name and platform of the load generator in the **Name** box, or select the load generator from the Load Generator dialog box.

4 Click **OK** to close the Load Generator Information or Add New Load Generator dialog box and save your settings.

Understanding the Load Generator Information Dialog Box

The Load Generator Information dialog box enables you to add a load generator to your scenario.

➤ **Name.** Type the name of the load generator you want to add in the Name box.

➤ **Platform.** Select the type of platform on which the load generator is running.

➤ **Temporary directory.** Type a location, on the load generator, where the Controller can store temporary files.

➤ **Enable load generator to take part in the scenario.** Select to include the load generator in the scenario.

Status Tab

In the Load Generator Information dialog box, select the **Status** tab to display details of the Load Generator Status.
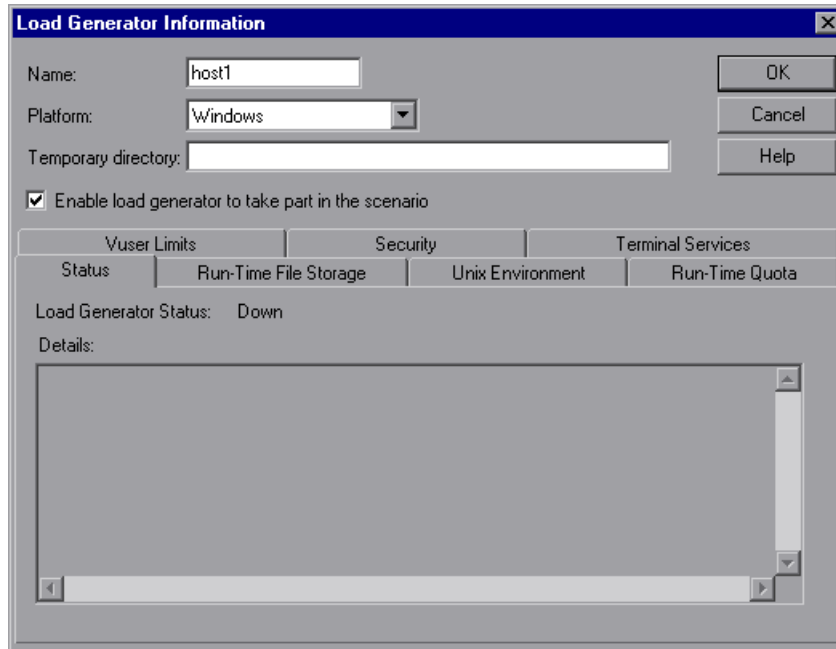
➤ **Load Generator Status.** Displays the status of the load generator.

➤ **Details.** Displays error and other run-time information about the selected load generator.

Run-Time File Storage Tab

In the Load Generator Information dialog box, select the **Run-Time File Storage** tab to specify the result directory for the performance data that LoadRunner gathers from each load generator during a scenario.
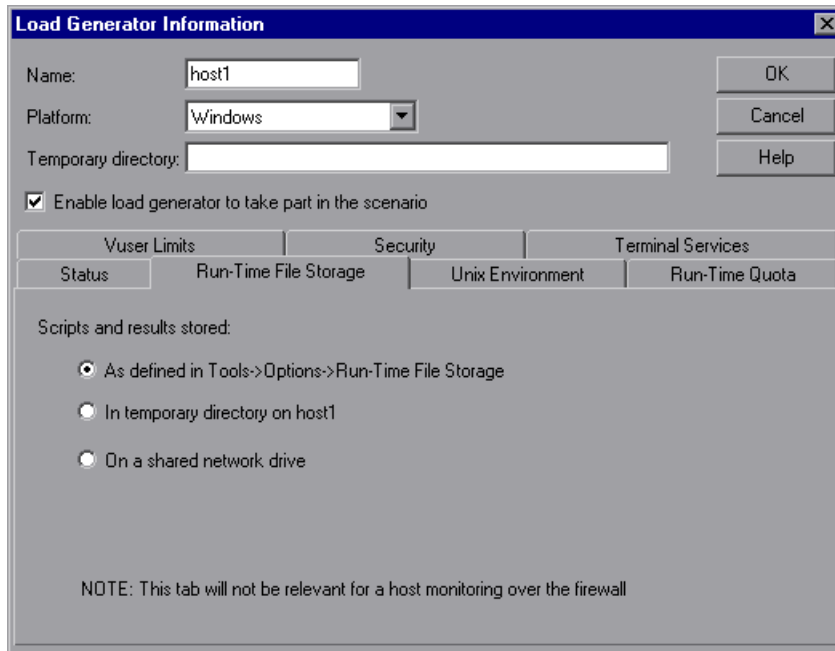
➤ **Scripts and results stored.** Select one of the following options:

  ➤ **As defined in Tools** > **Options** > **Run-Time File Storage.** Stores the results as specified in the global settings.

  ➤ **In temporary directory on <load generator name>.** Instructs the Controller to save the run-time files (results of the scenario run and Vuser scripts) on a hard drive of the load generator computer.

  ➤ **On a shared network drive.** Instructs the Controller to save the scenario results and/or the Vuser scripts on a shared network drive. A shared network drive is a drive to which the Controller and all the load generators in the scenario have read and write permission.

Note:

➤ If the load generator is **localhost**, then LoadRunner stores the scripts and results on a shared network drive, and the options for setting the location are all disabled.

➤ If you are monitoring over the firewall, the Run-Time File Storage settings are not relevant.

For information about setting the network location for the results, see Chapter 12, "Preparing to Run a Scenario."

UNIX Environment Tab

In the Load Generator Information dialog box, select the **UNIX Environment** tab to configure the login parameters and shell type for each UNIX load generator.
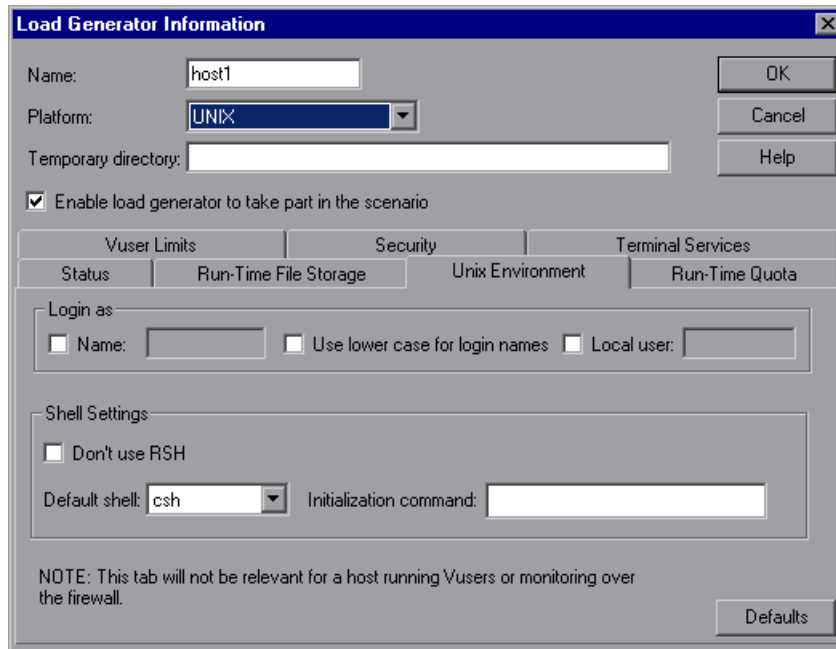
➤ **Login as.**

    ➤ **Name.** If the load generator is UNIX-based, set the login information for the load generator. By default, LoadRunner uses your NT user name for the UNIX login. In other words, if your NT login is **lrunner**, the Controller will log on to the load generator as **lrunner**. To log on to a UNIX-based load generator using a different login name, select the **Name** check box and specify the desired UNIX login name. Using this option, you can log on to the NT Controller as **bill** and connect to the UNIX load generator as **mike**. However, you should make sure that **mike** allows **bill** to log on using his name. This can be done by adding the line "+ bill" at the beginning of mike's **.rhosts** file.

    ➤ **Use lower case for login names.** Instructs LoadRunner to use lower case names during login to avoid case-sensitive issues with the UNIX operation system.

    ➤ **Local User** (Expert mode only). For information on this setting available in Expert mode, see Appendix C, "Working in Expert Mode."

➤ **Shell Settings.** Specify the UNIX shell settings for the remote UNIX load generator.

    ➤ **Don't use RSH.** Select this check box to connect to the remote load generator without using RSH. In this case, you need to activate the agent daemon on the load generator manually.

    To connect using RSH, clear this check box, and ensure that:

        ➤ RSH is enabled on the load generator (see the system configuration)

        ➤ the agent daemon is not already running on the load generator.

➤ **Default shell.** Select the default shell on the UNIX load generator: **csh** (C Shell—the default), **bsh** (Bourne Shell), or **ksh** (Korn Shell).

Note: To allow LoadRunner to run your application under the Korn shell, you first need to make sure that the **.profile** file contains all of the LoadRunner environment settings—for example, the M_LROOT definition and the LicenseManager variable. Your UNIX $M_LROOT/templates directory contains a template for the **.profile** file, called **dot profile**. Use the template as a guide for modifying your **.profile** file with the LoadRunner environment settings.

In addition, if you are using a Korn shell (ksh), you must delete all LoadRunner settings from the **.cshrc** file (e.g. M_LROOT) before executing the scenario.

➤ **Initialization command.** Enter any command line options for LoadRunner to use when logging on to a UNIX system. This initialization command will run as soon as the shell opens. For example, you could select **ksh** and use the following initialization command:
. .profile;

Note: If you are monitoring or running Vusers over the firewall, the UNIX Environment settings are not relevant.

Run-Time Quota Tab

Initializing or stopping a large number of Vusers simultaneously places large stress on a load generator. To reduce stress on a load generator, you can initialize or stop smaller batches of Vusers.

In the Load Generator Information dialog box, select the **Run-Time Quota** tab to specify the maximum number of Vuser types that the load generator will initialize or stop simultaneously.
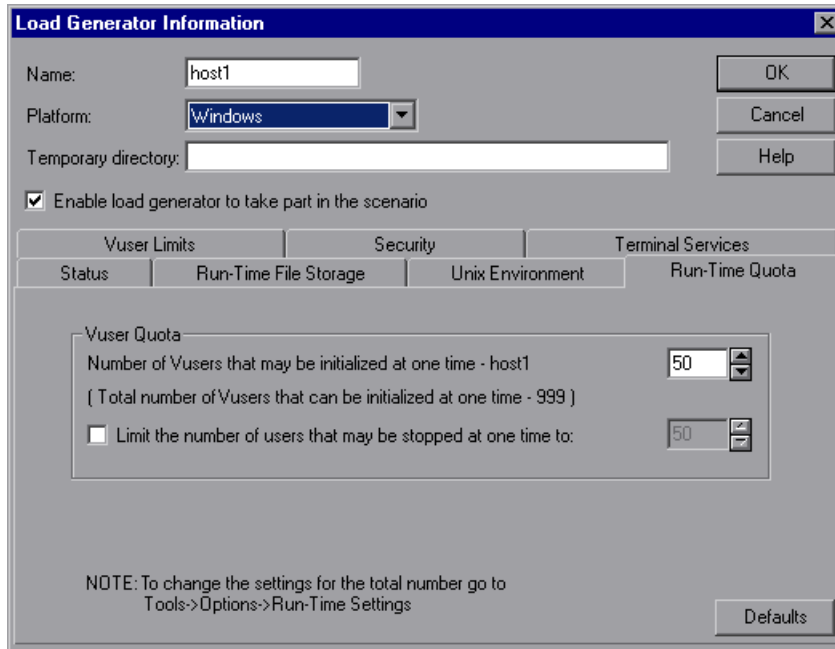
➤ **Vuser Quota**

  ➤ **Number of Vusers that may be initialized at one time - <current load generator>.** Select the maximum number of Vusers that the current load generator can initialize simultaneously.

  ➤ **Limit the number of users that may be stopped at one time to.** Select the maximum number of Vusers that the current load generator can stop simultaneously.

➤ **Defaults.** Sets the number of Vusers that may be initialized or stopped at one time to 50.

You can set run-time quotas for an entire scenario using the Run-Time Settings tab in the Options dialog box. For information on setting quotas globally for an entire scenario, see Chapter 11, "Configuring a Scenario."

Vuser Limits Tab

In the Load Generator Information dialog box, select the **Vuser Limits** tab to modify the maximum number of GUI, RTE, and other Vusers that a load generator can run.

➤ **Available Types.** Select the types of Vusers you want the load generator to run.

➤ **Maximum Active.** Select the maximum number of Vusers of each type for the load generator to run.

➤ **Defaults.** Sets GUI-WinRunner to 1, RTE to 1000, and Other Vusers to 5000.

Note: The maximum number of active Vusers that you specify must not exceed the number of Vusers that you are licensed to run. To check your Vuser licensing limitations, in the HP LoadRunner launcher window (**Start** > **Programs** > **LoadRunner** > **LoadRunner**), select **Configuration** > **LoadRunner License**.

Security Tab

In the Load Generator Information dialog box, select the **Security** tab to enable monitoring or running Vusers over a firewall.
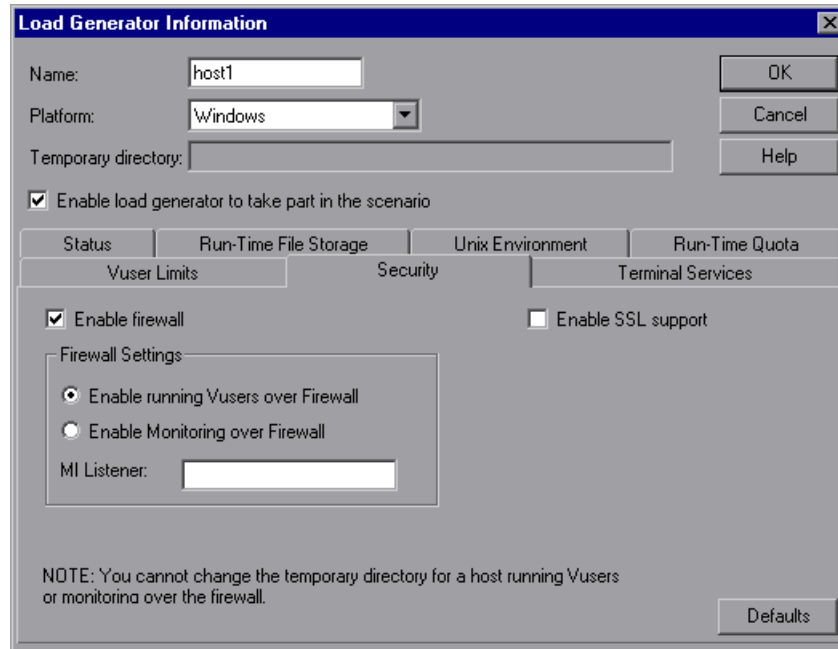
➤ **Enable firewall.** Enables LoadRunner to monitor or run Vusers over a firewall.

Note: If you select the **Enable firewall** option, the Temporary directory option for storing temporary files is disabled. Any location in the Temporary directory box is erased.

➤ **Enable SSL Support.** Enables an encrypted connection between the Controller and load generator.

➤ **Firewall Settings.**

➤ **Enable running Vusers over Firewall**. Enables LoadRunner to run Vusers on a load generator outside the firewall.

➤ **Enable Monitoring over Firewall.** Enables LoadRunner to monitor the load generator through a firewall.

➤ **MI Listener.** Type the name of the MI listener the load generator is using.

Note: If the load generator is connected, you cannot change values in the Security tab. To disconnect a load generator, select the load generator in the Load Generators dialog box and click **Disconnect**. The load generator status changes to **Down**, and you can change the settings.

The Security tab is disabled if the load generator is **Localhost**.

Vuser Status Tab

In the Load Generator Information dialog box, select the **Vuser Status** tab to view the status of all the Vusers connected to the selected load generator.
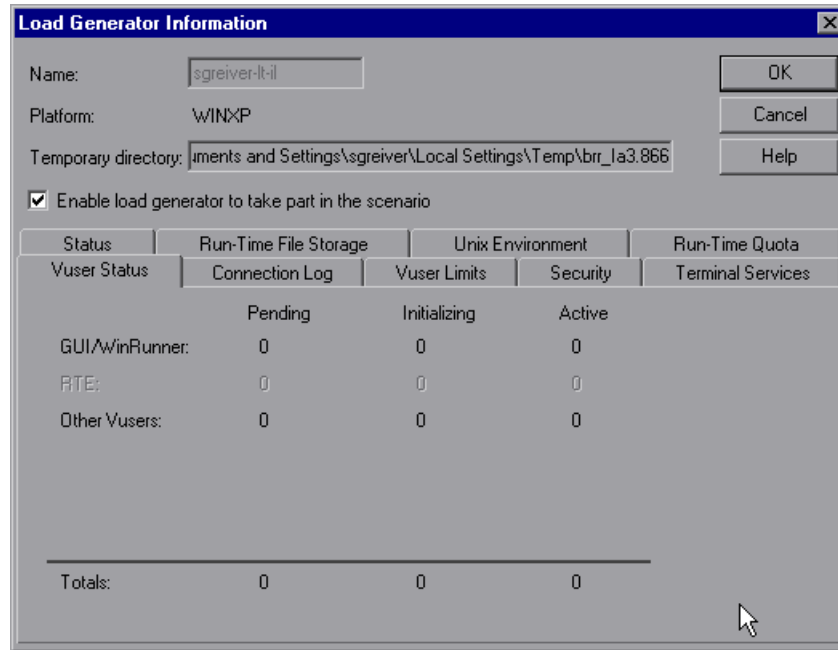
Note: This tab can only be viewed when the load generator is connected.

➤ **GUI/WinRunner.** Displays the number of GUI/WinRunner Vusers that are **Pending**, **Initializing**, and **Active**.

➤ **RTE.** Displays the number of RTE Vusers that are **Pending**, **Initializing**, and **Active**.

➤ **Other Vusers.** Displays the number of Vusers—other than GUI/WinRunner and RTE Vusers—that are **Pending**, **Initializing**, and **Active**.

➤ **Totals.** Displays the total number of Vusers that are **Pending**, **Initializing**, and **Active**.

Configuring Terminal Services Settings

You can use LoadRunner's Terminal Services Manager to remotely manage multiple load generators running in your load testing scenario on a terminal server. In addition, you can use a terminal server to overcome the limitation of being able to run only a single GUI Vuser on a Windows-based load generator. By opening a terminal server session for each GUI Vuser, you can run multiple GUI Vusers on the same application.

About Terminal Services

Terminal services allows centralized management of computing resources for each client connected to the server, and provides each user with their own working environment. Using a terminal server client, you can operate in a server-based computing environment from a remote machine. The terminal server transmits applications over the network and displays them via terminal emulation software. Each user logs on and sees only his individual session, which is managed transparently by the server operating system, independent of any other client session.

Important: Only users with administrator privileges can connect from the Controller to a local load generator via a terminal services session.

Examine the following diagram to understand how the LoadRunner components work together during a terminal session.
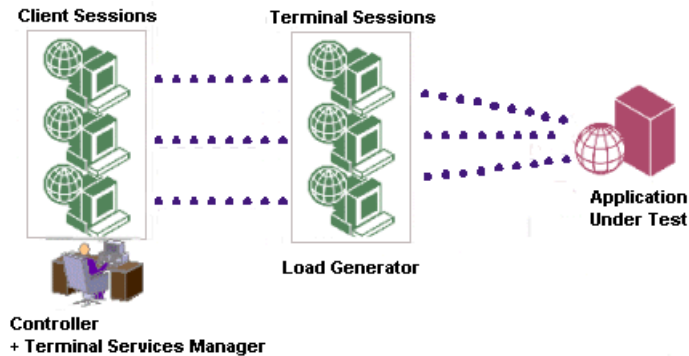
Terminal Services Manager

A terminal server client can have multiple terminal sessions running simultaneously. Using LoadRunner's Terminal Services Manager, you can select the number of terminals to be used in your scenario (provided that you have sufficient terminal sessions running), and the maximum number of Vusers that can be run per terminal. The Terminal Services Manager then evenly distributes the number of virtual users among the client sessions.

To use LoadRunner's Terminal Services Manager:

1 **Set up the terminal server agent on the Load Generator machine.**

2 **Launch a terminal client session on the Controller machine.**

3 **Distribute Vusers on the terminal server using the LoadRunner Terminal Services Manager.**

Setting up the Terminal Server Agent

Before you set up the terminal server agent on the load generator, make sure that a Load Generator has been installed on the Terminal server machine. For more information, see the *HP LoadRunner Installation Guide*.

To set up the Terminal Server Agent on the load generator:

1 Stop the LoadRunner agent by right-clicking its icon in the system tray and selecting **Close**.

2 Run **Agent Configuration** from **Start** > **Programs** > **LoadRunner** > **Advanced Settings**, or run **<LR>\launch_service\bin\AgentConfig.exe**. The Agent Configuration dialog box opens.
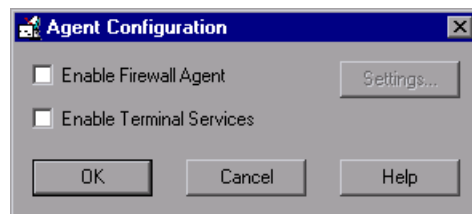
3 If you are running or monitoring Vusers over a firewall, select the **Enable Firewall Agent** check box, and then click **Settings**. See "Agent Configuration Settings" on page 321 for information on configuring the Agent Over Firewall settings.

4 Select the **Enable Terminal Services** check box.

Click **OK**.

5 Restart the LoadRunner agent as a process by double-clicking the shortcut on the desktop, or from **Start** > **Programs** > **LoadRunner** > **LoadRunner Agent Process**.

For more information about the LoadRunner Agent, see "Working with the LoadRunner Agent" on page 485.

Launching a Terminal Client Session

A terminal services client must be installed on the Controller machine before you can launch a terminal client session. For more information on installing a terminal services client, see the installation documentation for your Terminal server.

To open a Terminal Client session on the Controller machine:

1 Select **Start** > **Programs** > **Terminal Services Client** > **Terminal Services Client**. The Terminal Services Client dialog box opens.
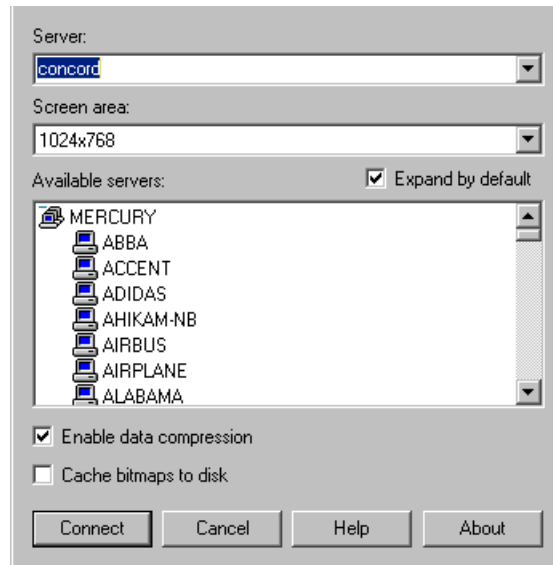
2 In the **Server** box, type the name or IP address of a terminal server, or select a terminal server from the list of available servers.

3 In the **Screen Area** box, select a window size for the terminal client.

4 Click **Connect**. The Windows logon dialog box opens.
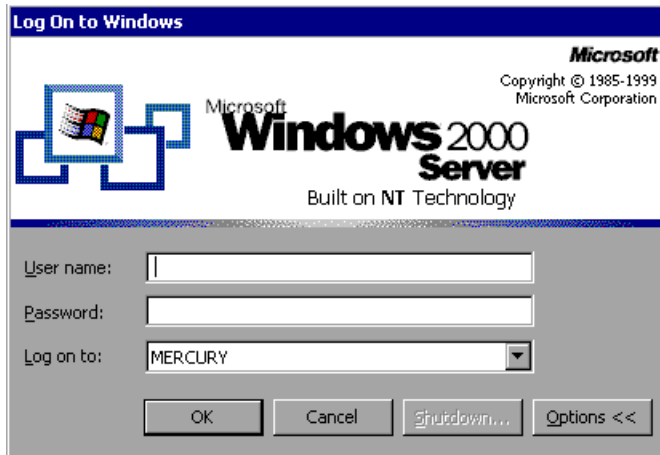
5 Enter your username, password, and domain name (if required) for the terminal server, and then click **OK**. A terminal client window opens.

6 Repeat steps 1 and 2 to open the number of sessions required.

Note: You must open a terminal server client session for each terminal that you want to run Vusers on during the scenario.

Distributing Vusers on the Terminal Server

In the Load Generator Information dialog box, select the **Terminal Services** tab to distribute Vusers running in your load testing scenario on a terminal server.

➤ **Enable Terminal Services Manager.** Enables the Controller to manage load automatically using terminal sessions on the load generator.

  ➤ **Number of terminals.** Enter the number of terminals that you want to use in your load test.

  ➤ **Maximum number of Vusers per terminal.** Enter the maximum number of Vusers that you want to run on each terminal. The maximum number of Vusers per terminal depends on the Vuser type used in the script.

Enabling this feature allows you to see the load generator's name only, without adding any extra references.

For example, if you need to use three load generator sessions, **my_machine**, **my_machine:1**, and **my_machine:2**, then in the load generator list, you need only insert the load generator, **my_machine**, and enable the Terminal Services Manager for 3 terminals.

Note: If this feature is not enabled, you need to insert each of the 3 terminals as separate load generators: **my_machine**, **my_machine:1**, and **my_machine:2**.

➤ **Connect to existing Terminal Services Sessions.** Enables connection to existing (open) terminal sessions.

Note: If you select this option, you must open a terminal client session manually for each terminal that you want to run Vusers on during the scenario.

➤ **Create new Terminal Services Sessions.** Enables the Controller to open and close terminal sessions automatically using a user name and password.

   ➤ **User name.** Enter a user name for the Terminal Services session.

   ➤ **Password.** Enter a password.

   ➤ **Domain.** Specify a domain for the Terminal Service session.

Note: This feature is not supported if the Controller and the load generators connect through a firewall.

➤ **Show Terminal Services clients on the Controller machine.** Select this option to enable interaction with new Terminal Services sessions using the RDP client.

➤ **Defaults.** Sets the number of terminals to 2, and the maximum number of Vusers per terminal to 50.

Troubleshooting

Check the connection between the Controller and the load generator on the terminal server. In the Controller, select the load generator in the Load Generators dialog box, and click **Connect**. The status changes from **Down** to **Ready** when the load generator is connected.

If there is no connection, check that the LoadRunner agent icon appears in the system tray of the terminal server. This indicates that the agent is running. Restart the LoadRunner agent if necessary from **Start** > **Programs** > **LoadRunner** > **LoadRunner Agent Service/Process**.

Configuring Scripts

Once you select a script for a Vuser or Vuser group, you can edit the script, or view the details of the script you selected, from the Vusers or Group Information dialog boxes.

To edit and view the details of a script used by a Vuser group:

1 Select the Vuser group whose script you want to modify.

2 On the Scenario Groups pane toolbar, click the **Details** button, or right-click the Vuser group and select **Details**. The Group Information dialog box opens displaying the current name and path of the script, and the type of script.

107
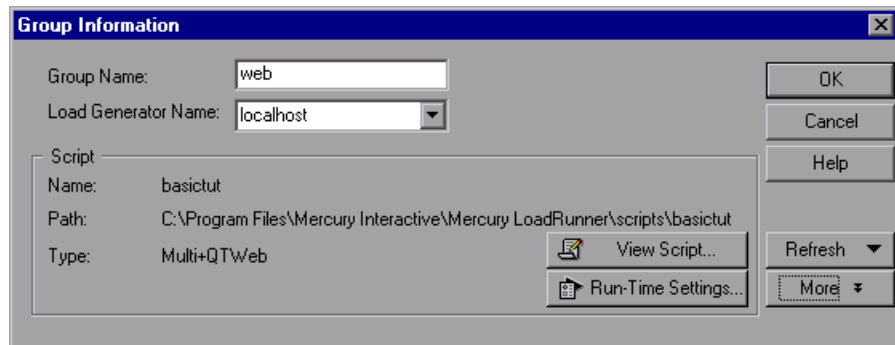
3 Click **Run-Time Settings** to set the script's run-time settings (optional). For details, see "Configuring Vuser Run-Time Settings" on page 81.

Note: If you modify the run-time settings from the Controller, LoadRunner runs the script using the modified settings. To restore the initial settings, click the **Refresh** button and select **Run-Time Settings**.

---

4 To edit the script, click **View Script**. The script generation tool, VuGen, opens. For more information on editing scripts, see the *HP Virtual User Generator User's Guide*.

Note: If you use VuGen to make changes to a script while the Controller is running, click the **Refresh** button and select **Script** to update the script details in the scenario.

---

5 Click **More** to expand the Group Information dialog box and view additional script information.
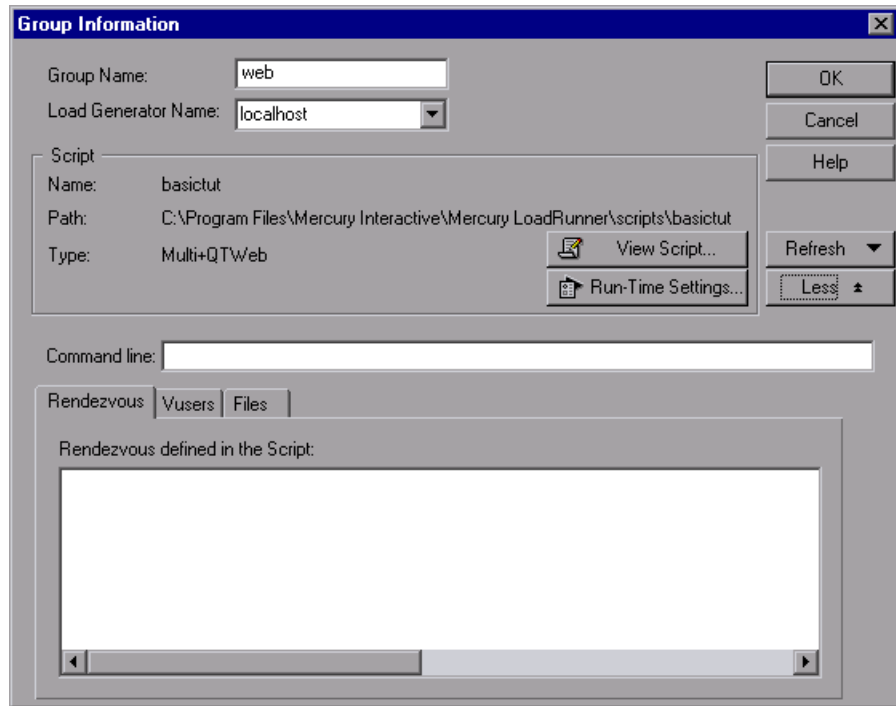
6 In the Command Line box, type any command line options to use when running the script. For example: -x value -y value

For information about passing command line argument values to a script, see the *HP Virtual User Generator User's Guide*.

7 To see the rendezvous points included in the selected script, click the **Rendezvous** tab.

8 To see the list of Vusers associated with the selected script, click the **Vusers** tab.
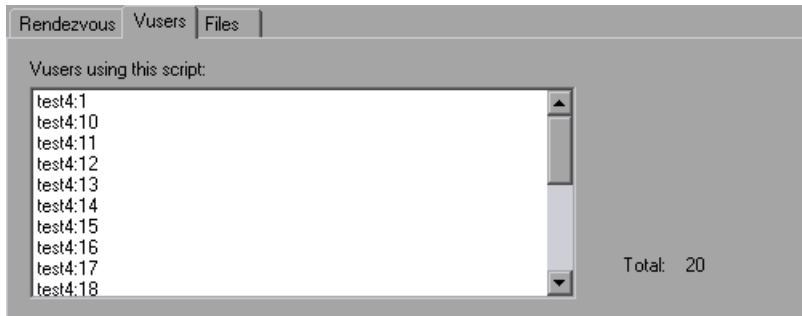
If you have not yet created Vusers, the box will be empty.

9 To see the list of files used by the script, click the **Files** tab.
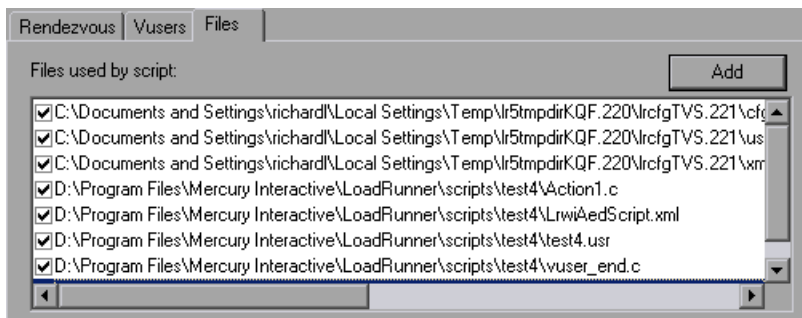
By default, this list shows all files in the script's directory (only after your script has been added to the script list). These files include the configuration settings file, the init, run, and end portions of the script, the parameterization definitions file, and the **.usr** file. To add a file to the list, click **Add** and add the file name. You can delete the files that you add, but not the other files listed.

Note: To run Visual C++ Vusers on a remote load generator, you must add the **.dll** of the Vuser to the Files used by script list.

---

10 Click **OK** to close the Group Information dialog box.

To edit and view the details of a script used by an individual Vuser:

1 On the Scenario Groups pane toolbar, click the **Virtual Users** button. The Vusers dialog box opens.
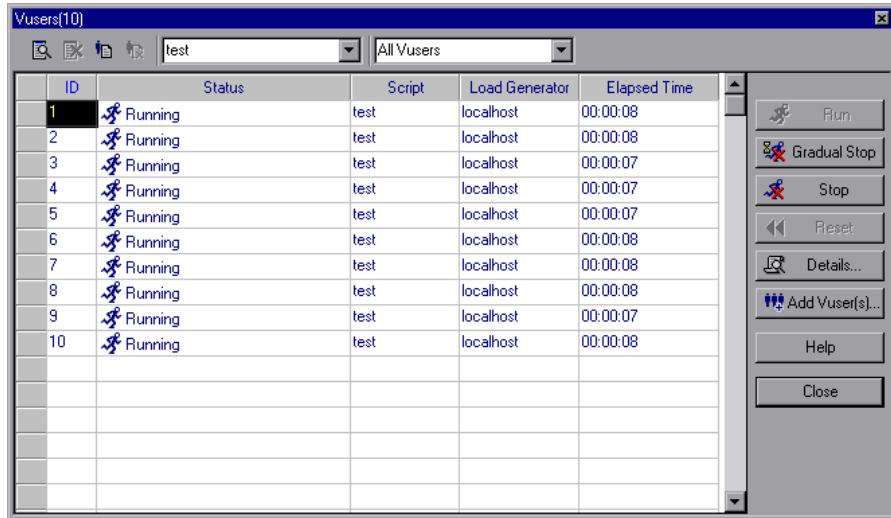
2 To view details of a script, click **Details**. The script's name and path are displayed in the Vuser Information dialog box. To select a different script, click the **Browse** button and select the path and file name of the new script. To select a VB Vuser script, browse to locate the **.usr** file.

Note: When you specify the location of a script, you can specify a location that is relative to the current scenario directory. For details, see "Using Relative Paths for Scripts" on page 112.

---

3 To edit a script, right-click the script in the Vusers dialog box, and select **View Script**. The script generation tool, VuGen, opens. For more information on editing scripts, see the *HP Virtual User Generator User's Guide*.

4 To modify the run-time settings you specified while recording a script using VuGen, right-click the script in the Vusers dialog box, and select **Run-Time Settings**.

Note: Modifying the run-time settings for one Vuser modifies the run-time settings for all the Vusers in the group that are using the same script.

If you highlight more than one script, you can modify the run-time settings in shared mode, as described in "Modifying Run-Time Settings for Multiple Scripts" on page 82.

For details about each run-time setting, see the *HP Virtual User Generator User's Guide*.

Using Relative Paths for Scripts

When you specify the location of a script, you can specify a relative location. The location can be relative to the current scenario directory, or the LoadRunner installation directory.

You can specify a path relative to the current scenario directory by typing either of the following notations at the start of the script path:

Notation	Description
.\	Indicates that the path is relative to the location of the scenario directory
..\	Indicates that the path is relative to the location of the parent directory of the scenario directory

For example, if the current scenario is located at F:\scenarios, to specify a script located at F:\scenarios\scripts\user1.usr, you could type:

```
.\scripts\user1.usr
```

You can specify a path relative to the LoadRunner installation directory by typing a percent sign (%) at the beginning of the script path. For example, if the LoadRunner installation directory is located at F:\LoadRunner, to specify a script located at F:\LoadRunner\scripts\user1.usr, you could type:

```
%\scripts\user1
```

Note: When specifying a relative path, you can include standard DOS notation (.\ and ..\) inside the path, as shown in the following example: M:\LR\my_tests\..\..\test.usr.

When you run a scenario, by default, the script is copied to a temporary directory on the Vuser group machine. This enables the Vuser group load generator to access the script locally instead of over a network.

You can instruct the Controller to store the script on a shared network drive (see Chapter 11, "Configuring a Scenario"). If you configure the Controller to save the script to a network drive, you must ensure that the Vuser load generator recognizes the drive. The Script window contains a list of all the Vuser scripts and their paths. A script's path is based on the Controller load generator's mapping of that location. If a Vuser load generator maps to the script's path differently, path translation is required. Path translation converts the Controller load generator's mapping to the Vuser load generator's mapping. For more information see Appendix B, "Performing Path Translation."

6

Creating a Manual Scenario Using the Percentage Mode

You build a manual scenario in the Percentage mode by defining the total number of Vusers to be used in the scenario, and assigning load generators and a percentage of the total number of Vusers to each script. This chapter describes how to create a manual scenario in the Percentage mode.

This chapter includes:

➤ About Creating a Manual Scenario Using the Percentage Mode on page 116

➤ Defining the Total Number of Vusers on page 118

➤ Assigning Properties to Scripts on page 119

➤ Configuring Scripts on page 122

➤ Converting a Scenario to the Vuser Group Mode on page 128

About Creating a Manual Scenario Using the Percentage Mode

When you design a regular manual scenario, you create Vuser groups, assign them scripts, load generators, and virtual users. When you work in the Percentage Mode, you define the total number of Vusers to be used in the scenario, and assign load generators and a percentage of the total number of Vusers to each script.

When you create a new scenario, you can access the Percentage Mode directly by selecting **Use the Percentage Mode to distribute the Vusers among the scripts** in the New Scenario dialog box. You can also convert a scenario created in the Vuser Group Mode to the Percentage Mode by choosing **Scenario** > **Convert Scenario to the Percentage Mode**.

When converting your scenario from Vuser Group Mode to Percentage Mode, note the following:

➤ If you defined multiple scripts for a Vuser group, the number of Vuser scripts created in the Percentage Mode will equal the number of scripts defined for the group.

➤ <All Load Generators> will be assigned to all Vuser scripts created in the Percentage Mode. If you defined multiple load generators for a Vuser group, the Vusers you assign to the scripts in the Percentage Mode will be distributed evenly among the load generators you previously assigned to the group.

➤ All Vuser group schedule settings will be lost. All profiles will contain scenario schedule settings only.

Understanding the Percentage Mode Design Tab

When you create a manual scenario using the percentage mode, the Controller's Design tab displays the Scenario Scripts, Service Level Agreement, and Scenario Schedule panes.

The **Scenario Scripts** pane lists all enabled and disabled Vuser scripts, their paths, the load generators, and the percentage of the total number of Vusers assigned to each script.

The **Scenario Schedule** pane displays information regarding the schedule profile: its name, the schedule mode, the duration of the scenario, the scenario run mode, the total number of Vusers to be used in the scenario, and the load behavior. The Interactive Schedule graph displays a graph of the scenario schedule you defined. For more information on creating a scenario schedule, see Chapter 7, "Scheduling Scenarios."

The **Service Level Agreement** pane lists all the service level agreements (SLAs) defined for the scenario. For more information on SLAs, see Chapter 8, "Defining Service Level Agreements."

You can perform the following actions on a Vuser script or scenario:

➤ Define the total number of Vusers to be used in the scenario

➤ Define the script name, script path, load generators, and percentage of the total number of Vusers for the Vuser script

➤ Add one or more load generators to the Vuser script and configure the load generators

➤ Add a new script to the scenario and configure it

➤ Enable or disable a Vuser script for the scenario

➤ Remove a Vuser script from the scenario

➤ Schedule the scenario

➤ Define service level agreements for the scenario

➤ Run the scenario

➤ Stop the scenario

➤ Reset the scenario

➤ Configure scenario result settings

Defining the Total Number of Vusers

When you build a scenario in the Percentage mode, you define the total number of Vusers to be used for the scenario when you create a schedule for the scenario. For information on creating a scenario schedule, see Chapter 7, "Scheduling Scenarios."
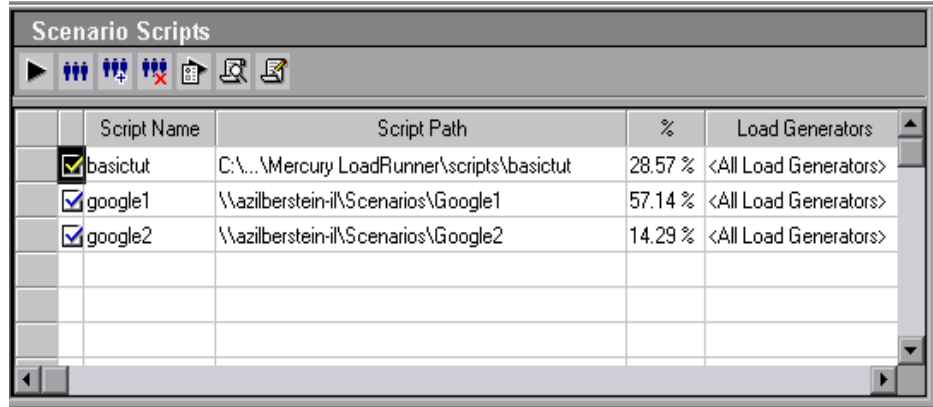
Note: The Vuser Group settings are not available for the Percentage mode.

Assigning Properties to Scripts

The Scenario Scripts pane displays a list of scripts you selected in the New
Scenario dialog box, or defined in the Vuser Group mode.

	Script Name	Script Path	%	Load Generators	
☑	basictut	C:\...\Mercury LoadRunner\scripts\basictut	28.57 %	<All Load Generators>	
☑	google1	\\azilberstein-il\Scenarios\Google1	57.14 %	<All Load Generators>	
☑	google2	\\azilberstein-il\Scenarios\Google2	14.29 %	<All Load Generators>	

The % column indicates the percentage of the total number of Vusers that is
automatically distributed to each Vuser script. During scenario execution,
each script runs the percentage of Vusers assigned to it. The Load Generators
column automatically contains <All Load Generators> for each Vuser script.

Note: If you defined multiple load generators for a Vuser group, the Vusers
you assign to the scripts in the Percentage Mode will be distributed evenly
among the load generators you previously assigned to the group.

For each script you can modify:

➤ the percentage of the total number of Vusers assigned to the script

➤ the load generators on which the Vusers will execute the script

To modify the percentage of Vusers assigned to a script:

In the script's % column, enter a percentage of the total number of Vusers you defined in scenario schedule. The percentages assigned to the other scripts change to create a total of 100 percent for all of the Vuser scripts.

To add or modify load generators for a script:

1 In the script's Load Generators column, select one or more load generators from the Load Generator Name list, and click **OK**. If you select multiple load generators, the Vusers assigned to the script are distributed evenly among the load generators.

2 Alternatively, to add a load generator to the list, select **<Add..>**. The Add New Load Generator dialog box opens.
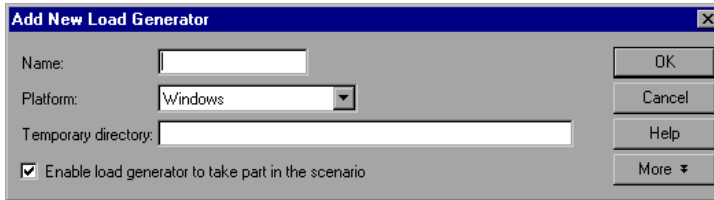
3 In the **Name** box, type the name of the load generator.

4 In the **Platform** list, select the type of platform on which the load generator is running.

5 In the **Temporary Directory** box, type a location on the load generator, where the Controller can store temporary files, or leave the box empty to accept the default location. By default, LoadRunner stores temporary files on the load generator during scenario execution, in a temporary directory specified by the load generator's TEMP or TMP environment variables.

6 To allow the load generator to take part in the scenario, check **Enable load generator to take part in the scenario**.

7 Click **More** to expand the dialog box and show the Add Load Generator tabs. For information on configuring settings for each load generator, see "Configuring Load Generator Settings" on page 88.

8 Click **OK** to close the Add Load Generator dialog box. LoadRunner adds the new load generator to the Load Generator Name list. To include the new load generator in your scenario, select it from the Load Generator Name list, and click **OK**.

Repeat the above procedure for each load generator you want to add to your scenario.

Note: The Controller monitors a Windows load generator's CPU usage and automatically stops loading Vusers on the overloaded load generator, and distributes them among other load generators taking part in the scenario. For more information, see "Load Balancing" on page 193.
You can monitor the status of a machine's CPU usage using the icons in the Load Generators dialog box. When the CPU usage of a load generator becomes problematic, the icon to the left of the load generator name contains a yellow bar. When the machine becomes overloaded, the icon contains a red bar.

Configuring Load Generators

You can set a load generator's attributes while adding it to the load generator list, or modify the attributes of an existing load generator at any time, using the Load Generators dialog box. You can also use the Load Generators dialog box to indicate which load generators will run Vusers in the scenario. For example, if a load generator is unavailable for a particular scenario run, you can use the Load Generators dialog box to exclude it temporarily instead of removing it entirely from your list of load generators. For instructions on using the Load Generators dialog box, see "Understanding the Load Generators Dialog Box" on page 85.

To configure global settings for all load generators participating in the scenario, use LoadRunner's Options dialog box. For more information, see Chapter 11, "Configuring a Scenario."

Configuring Scripts

You can add a script to the Scenario Scripts list using the Add Script dialog box. Once you have added the script to the list, you can view the details of the script you selected, edit the script, enable/disable it, or change its run-time settings.

To add a script:

1 On the Scenario Scripts pane toolbar, click the **Add Scripts** button, or right-click within a column and select **Add Script**. The Add Script dialog box opens.
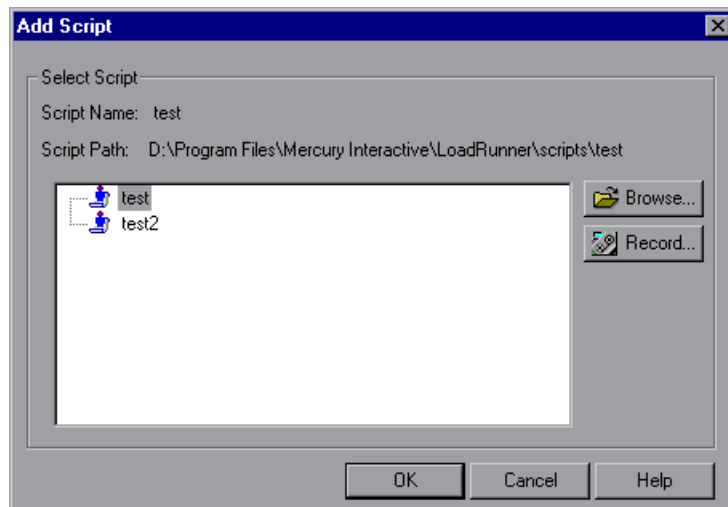
2 Click **Browse**. The Open Test dialog box opens.

Select the path and file name of the new script.

Note: When you specify the location of a script, you can specify a location that is relative to the current scenario directory. For details, see "Using Relative Paths for Scripts" on page 112.

---

3 Click **Open** to select the files. The Open Test dialog box closes, and the new script name appears in the Add Script dialog box.

4 Click **OK** to close the Add Script dialog box and enter the new script information in the Scenario Scripts window.

Understanding the Add Script Dialog Box

The Add Script dialog box enables you to add scripts to your scenario.

➤ **Select Script.** Displays the available scripts in the current directory.

> ➤ **Script Name.** Click the script you want to add to your scenario. The script appears in the Script Name column.

> ➤ **Script Path.** Displays the script directory's path.

> ➤ **Browse.** Select a script from a different directory. To select a VB Vuser script, browse to locate the **.usr** file.

> ➤ **Record.** Opens the Virtual User Generator so that you can begin recording a script. For more information on recording scripts, see the *HP Virtual User Generator User's Guide*.

Note: While a scenario is running, you can add Vuser scripts to the scenario and enable them. However, if you add a script after all the Vusers in the scenario have been ramped up, the new script will not run in the scenario.

Viewing Script Information

Once you have added a script to the list, you can view the details of the script you selected, edit the script, enable/disable it, or change its run-time settings.

To view script details:

1 Select the script.

2 On the Scenario Scripts pane toolbar, click the **Details** button, or right-click the script and select **Details**. The Script Information dialog box opens, displaying the path, name, and type of the script you selected.
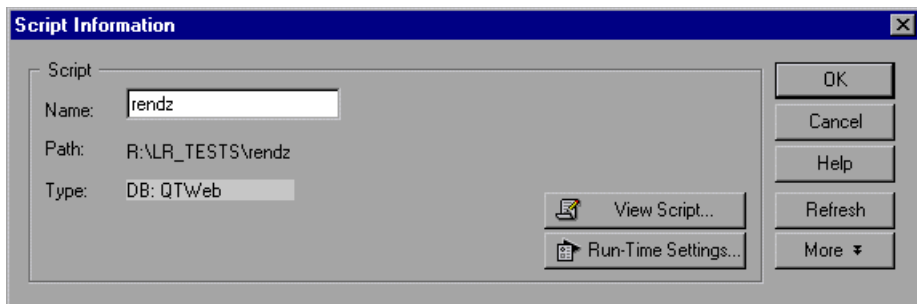
3 Click **Run-Time Settings** to set the script's run-time settings (optional), which allows you to customize the way the Controller executes a Vuser script. The Run-Time Settings dialog box opens, displaying the settings you previously set using VuGen.

Note: If you modify the run-time settings from the Controller, LoadRunner runs the script using the modified settings. To restore the initial settings, click **Refresh** and select **Run-Time Settings**.

---

4 To edit the script, click **View Script**. The script generation tool, VuGen, opens. For more information on editing scripts, see the *HP Virtual User Generator User's Guide*.

Note: If you use VuGen to make changes to a script while the Controller is running, click **Refresh** and select **Script** to update the script details in the scenario.

---

5 Click **More** to expand the Script Information dialog box and view additional script information.

6 In the Command Line box, type any command line options to use when running the script. For example, -x value -y value

For information about passing command line argument values to a script, see the *HP Virtual User Generator User's Guide*.

7 To see the rendezvous points included in the selected script, click the **Rendezvous** tab.

8 To see the list of Vusers associated with the selected script, click the **Vusers** tab.

9 To see the list of files used by the script, click the **Files** tab. By default this list shows all files in the script's directory (only after your script has been added to the script list). These files include the configuration settings file, the init, run, and end portions of the script, the parameterization definitions file, and the **.usr** file. To add a file to the list, click **Add** and add the file name.

Note: You can delete the files that you add, but not the other files listed.

---

10 Click **OK** to close the Script Information dialog box.

To delete a script:

1 Select the script.

2 On the Scenario Scripts pane toolbar, click the **Remove Script** button, or right-click the script and select **Remove Script**.

To disable a script:

Clear the box to the left of the Vuser script name. The color of the script entry changes to gray, indicating that the script will not take part in the scenario. To enable the Vuser script, select the check box again.

Understanding the Script Information Dialog Box

Lets you view the details of a selected script and modify its settings.

➤ **Script.** Displays details of the selected script.

➤ **Name.** Displays the name of the selected script. To modify the name, type the modified name in the Name box.

➤ **Path.** Displays the script directory's path.

➤ **Type.** Displays the type of the selected script.

➤ **View Script.** Opens the Virtual User Generator, so that you can edit the script. For more information on editing scripts, see the *HP Virtual User Generator User's Guide*.

➤ **Run-Time Settings.** Opens the Run-Time Settings dialog box, enabling you to edit the script run-time settings you previously set using VuGen. If you did not set run-time settings for a script in VuGen, the default VuGen settings are displayed for all but the Log and Think Time tabs, which display the default Controller settings. Note that several protocols, such as Web and Java, have specific settings. For information on the run-time settings, see the *HP Virtual User Generator User's Guide*.

➤ **Refresh.** Click this button and select **Script** to update the script details in the scenario, if you make any changes to a script while the Controller is running. Select **Run-Time Settings** to restore the initial run-time settings, if you modify the run-time settings from the Controller.

➤ **More/Less.** Reveals/hides the following:

➤ **Command Line.** Type any command line options to use when running the script—for example: -x value -y value. For information about passing command line argument values to a script, see the *HP Virtual User Generator User's Guide*.

➤ **Rendezvous.** Displays the rendezvous points defined for the selected script.

➤ **Vusers.** Displays all Vusers associated with the selected script.

➤ **Files.** Displays all files used by the script. To exclude a file from the list, select the check box adjacent to it. To add a file to the list, click **Add**.

Converting a Scenario to the Vuser Group Mode

You can convert a scenario created in the Percentage Mode to the Vuser Group Mode by choosing **Scenario** > **Convert Scenario to the Vuser Group Mode**.

Note: You can also convert a scenario in the Vuser Group Mode to the Percentage Mode. For more information, see "About Creating a Manual Scenario Using the Percentage Mode" on page 116.

LoadRunner displays a warning that you are about to convert your Manual scenario from Vuser Group Mode to Percentage Mode, or vice versa. If you want to convert your scenario, click **Yes**. If you want to remain in the current mode, click **No**.

➤ **Always show this dialog box before converting the scenario.** Clear this box if you do not want to be prompted with the current warning. To restore the current warning, select **Scenario** > **Show Convert Scenario Warning**.

When converting your scenario from Percentage Mode to Vuser Group Mode, note the following:

➤ Each script will be converted to a Vuser group.

➤ If you defined multiple load generators for a Vuser script, the Vuser group that is created when converting the scenario will also contain multiple load generators.

➤ All schedule settings will be kept.

7

Scheduling Scenarios

After you create a manual scenario, you define a schedule according to which the scenario should run. You can schedule when to initialize, start, and stop Vusers during the scenario run, and how long an action should continue running.

This chapter includes:

About Scheduling Scenarios

An important factor in the creation of a scenario is developing a test that accurately portrays user load behavior—the type of load and the timing of the load.

After you create a scenario, you schedule the scenario to start running at a specified time. You can limit the execution duration of the scenario or of a Vuser group within the scenario.

You can also stipulate how many Vusers to start and stop running within a certain time frame. You can specify whether LoadRunner should start or stop running all Vusers in a scenario simultaneously, or only a certain number of Vusers within a specified amount of time.

Note: Rendezvous points in a Vuser script interfere with a scheduled scenario run. If your script contains rendezvous points, your scenario will not run as scheduled.

Scheduler at a Glance

You configure a schedule for a manual scenario in the Design tab, in the Scenario Schedule pane.
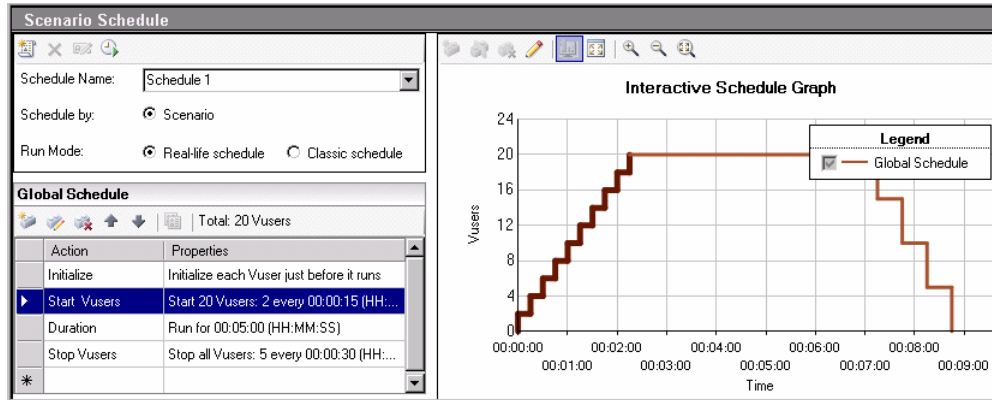
The Scenario Schedule pane contains the following sections:

Schedule Definition Area

The Schedule Definition area displays a selected schedule, the schedule type, and its run mode. In this area, you can create a schedule, select an existing schedule to modify, rename a schedule, or delete a schedule.

For more information, see "Managing Scenario Schedules" on page 139.

Schedule Actions Grid

You define the actions for a schedule in the Actions grid. You can add, modify, and delete actions. These actions include initializing, starting, and stopping Vusers, as well as defining how long to continue running the actions.

When creating group schedules, you can copy a Vuser group's schedule settings to other Vuser groups.

For more information about schedule actions, see "Understanding Schedule Actions" on page 135.

Interactive Schedule Graph

The interactive schedule graph displays a graphical representation of the scenario schedule. The lines in the graph correspond to the actions defined in the Actions grid. The graph is dynamic—that is, you can modify the schedule actions from the graph itself.

For more information, see "Viewing Scenario Schedules in the Interactive Graph" on page 152.

Scheduling by Scenario or Group

After you have created a scenario, you can schedule the enabled Vuser groups to run as part of a whole scenario or you can schedule each Vuser group to run on its own schedule.

For information on designing a manual scenario, see Chapter 5, "Creating Manual Scenarios."

Scheduling By Scenario

When you run a scenario, LoadRunner runs all the Vuser groups enabled in the scenario. The schedule defined for running the scenario is applied to all the Vuser groups concurrently, and LoadRunner applies each action proportionately to all the Vusers groups.

For example, take a scenario that includes 3 enabled Vuser groups, where Group1 contains 10 Vusers, Group2 contains 20 Vusers, and Group3 contains 30 Vusers, giving a total of 60 Vusers. When scheduling by scenario, if the schedule instructs LoadRunner to load 30 Vusers when it starts running, LoadRunner loads a proportional number of Vusers from each group—that is, 5 from Group1, 10 from Group2, and 15 from Group3, totalling 30 Vusers.

The same principle applies when viewing the scenario in Percentage mode.

Scheduling By Vuser Group

For each enabled Vuser group in a scenario, you can design a separate execution schedule. You can specify when to start running the Vuser group, how many Vusers to start and stop running within given time intervals, and how long the Vuser group should continue running.

Note: You cannot view Group schedules when viewing a scenario in Percentage mode.

Schedule Run Modes

You can schedule a scenario to run according to the run-time settings defined in the enabled Vuser groups, or you can let the groups run over and over again until the scenario schedule instructs them to stop running.

You can schedule a scenario to run in one of the following modes:

➤ **Real-life schedule.** The scenario runs according to a user-defined group of actions that simulate a real-life schedule of events. Vuser groups run according to the iterations defined in their run-time settings, but you can define how many Vusers to run at a time, how long Vusers should continue to run, and how many Vusers to stop running at a time.

➤ **Classic Schedule.** All enabled Vuser groups run together on one schedule, each according to its own run-time settings. You can schedule how many Vusers to start running at a time, how long to run the Vusers, and how many Vusers to stop running at a time.

➤ **Run until complete.** All the Vuser groups in the scenario run according to the iterations defined in their run-time settings. Each Vuser group in the scenario runs its defined course, and when all the Vuser groups have finished running, the scenario run is complete.

Configuring the Default Scheduler Run Mode

When you create a scenario for the first time, the default run mode for the Scheduler is one that emulates a real-life set of events, **Real-life schedule**. If you want to create classic schedules, you can set the default run mode to **Classic schedule**.

To set the default Scheduler run mode:

1 In the Controller, select **Tools** > **Options**.

2 In the Options dialog box, click the **Scheduler** tab.
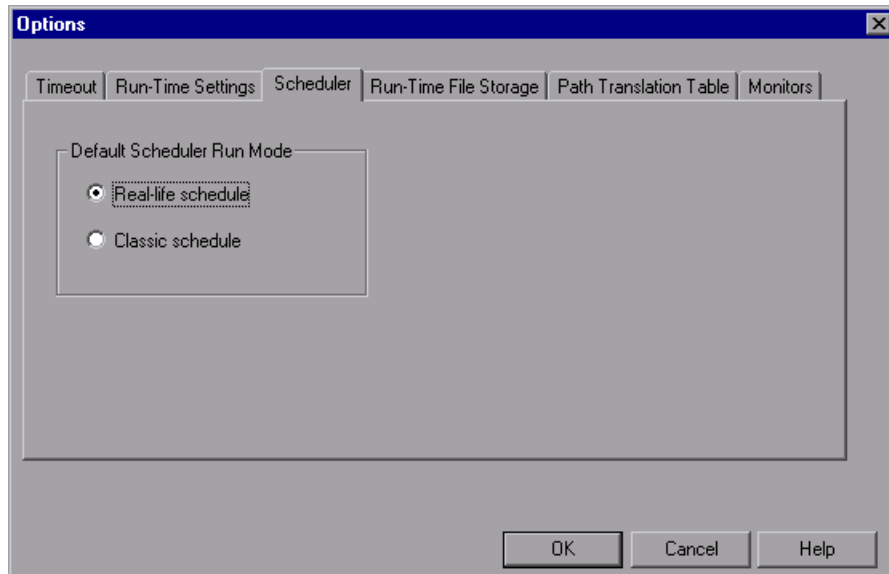
3 Select a default Scheduler run mode, and click **OK**.

By default, all new schedules will open in the selected mode.

Understanding Schedule Actions

A scenario schedule contains a series of actions that instruct the scenario when to start running a Vuser group, when to initialize, start, and stop the Vusers, and how long to run an action.

The following sections describe the available schedule actions.

For information on managing the schedule actions, see "Managing Schedule Actions" on page 146.

Start Group

The Start Group action defines when to start running a Vuser group. This action is available for group schedules only.

Note: By default, the Start Group action appears as the first action in the Actions grid when you select **Schedule by: Group**. It is always followed by the **Initialize** action. It cannot be deleted.

Options	Description
Start immediately after the scenario begins (Default)	LoadRunner starts running the selected Vuser group as soon as the scenario starts running. (See "Scheduling a Scenario Start Time" on page 144.)
Start <00:00:00> (HH:MM:SS) after the scenario begins	After the scenario starts running, LoadRunner waits the specified time (in hours, minutes, and seconds), before it starts running the selected Vuser group.
Start when group <group name> finishes	LoadRunner starts running the selected Vuser group immediately after the Vuser group specified in this option has finished running.

Initialize

The Initialize action instructs LoadRunner to prepare the Vusers so that they are in the Ready state and can run.

Note: By default, the Initialize action appears in the Actions grid for all schedule types. It cannot be deleted.

Options	Description
Initialize all Vusers simultaneously	LoadRunner initializes all the Vusers in the scenario or selected Vuser group together, before running them.
Initialize XX **Vusers every** <00:00:00> **(HH:MM:SS)**	LoadRunner initializes the specified number of Vusers gradually before running them, according to the specified time interval (in hours, minutes, and seconds).
Initialize each Vuser just before it runs (Default)	LoadRunner initializes the each Vuser in the scenario or selected Vuser group just before it starts running. **Note:** This option is not available for Group schedules when the **Wait for all groups to initialize** option is selected. See "Initializing All Vuser Groups Before a Run" below.

Initializing All Vuser Groups Before a Run

When creating a schedule by group, you may want the Scheduler to wait until all the Vusers in all the groups have finished initializing, before starting to run any of the groups.

If you select this option, if any of the Vuser groups in the scenario are set to **Initialize each Vuser just before it runs**, the Scheduler automatically changes this setting to **Initialize all Vusers simultaneously**.

For more information, see "Initializing All Vuser Groups Before a Run" on page 143.

Start Vusers

The Start Vusers action instructs LoadRunner to start running Vusers.

Options	Description
Start XX **Vusers: Simultaneously** (Default)	LoadRunner runs the specified number of Vusers simultaneously.
Start XX **Vusers:** YY **Vusers every** <00:00:00> **(HH:MM:SS)**	LoadRunner runs the specified number of Vusers (XX) gradually. That is, LoadRunner runs YY Vusers, and waits the specified time (in hours, minutes, and seconds) before running another YY Vusers.

Note:

➤ LoadRunner starts running Vusers only when they have reached the Ready state.

➤ In a Classic schedule, LoadRunner always runs **all** the Vusers, whether simultaneously or gradually. In a Real-life schedule, you can select how many Vusers to run.

➤ While a scenario is running, you can add Vuser groups/scripts to a scenario and enable them. When starting the Vusers gradually, if you add a Vuser group after all the Vusers in the scenario have started running, the new group starts running immediately.

Duration

The Duration action instructs LoadRunner to continue running the scenario in the current state, for the specified amount of time.

Options	Description
Run until completion (Classic schedule only)	The scenario runs until all the Vusers have finished running.
Run for XX days and <00:00:00> **(HH:MM:SS)**	The scenario runs in its current state for the specified amount of time (in days, hours, minutes, and seconds) before continuing with the next action. The default Duration period is 5 minutes.
Run indefinitely (Classic schedule only)	The scenario runs indefinitely.

Note: In a Real-life schedule, if a Duration action is not followed by another action, the scenario continues to run indefinitely.

Stop Vusers

The Stop Vusers action instructs LoadRunner to stop the running Vusers.

Options	Description
Stop XX **Vusers: Simultaneously** (Default)	LoadRunner stops the specified number (**All** or XX) of running Vusers at once.
Stop XX **Vuser:** YY **Vusers every** <00:00:00> **(HH:MM:SS)**	LoadRunner stops the specified number of Vusers (**All** or XX) gradually. That is, LoadRunner stops YY Vusers, and waits the specified time (in hours, minutes, and seconds) before stopping another YY Vusers.

Managing Scenario Schedules

You create and modify scenario schedules in the Schedule Definition area of the Scheduler.
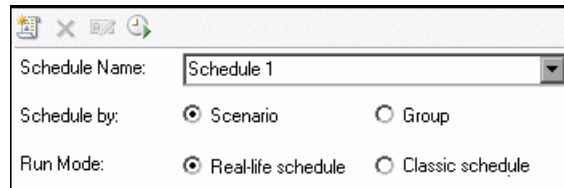
This section includes:

Schedule Definition Toolbar

Toolbar Button	Button Name	Enables you to
	New Schedule	Create a schedule
	Delete Schedule	Delete a selected schedule
	Save New Name	Rename a selected schedule
	Start Time	Set the start time for a scenario
	Wait for all groups to finish initialization	(Schedule by Group only) Set all the Vuser groups in the scenario to finish initializing before running any of the Vusers.

Creating Scenario Schedules

You create a scenario schedule in the Schedule Definition area. When you create a new schedule you choose the type of schedule, select a run mode, and select a set of actions for the schedule.

To create a schedule:

1 Define the schedule's properties.

a In the Schedule Definition area, click the **New Schedule** button. The **Schedule Name** box displays a new schedule, **Schedule <x>**.

(Optional) You can rename the schedule by typing a different name, and clicking the **Save New Name** button.

b Select a type of schedule and run mode as follows:

Note: When switching from one run mode to another, all existing actions for the schedule are replaced by the default set of actions for the new run mode.

Schedule by	Run Mode	
Scenario	**Real-life schedule.** All enabled Vuser groups run together on one schedule. The scenario runs according to a user-defined group of actions that emulate a true-to-life schedule of events. You can schedule how many Vusers to start running at a time, how long to run the Vusers, and how many Vusers to stop running at a time.	**Classic Schedule.** All enabled Vuser groups run together on one schedule, each according to its own run-time settings. You can schedule how many Vusers to start running at a time, how long to run the Vusers, and how many Vusers to stop running at a time.
Group (Not applicable when viewing scenario in Percentage mode)	**Real-life schedule.** Each enabled Vuser group runs according to its own defined schedule that emulates a true-to-life schedule of events for that Vuser group. You can schedule when to start running the Vuser group, how many Vusers to run at a time, how long to run the Vusers, and how many Vusers to stop running at a time.	**Run until complete.** Each enabled Vuser group runs according to its own schedule, each according to its own run-time settings. For each Vuser group, you can schedule how many Vusers to start running at a time. When all the Vuser groups have finished running, the scenario run is complete.

2 Define a set of actions for the schedule.

The Actions grid displays the default actions that correspond to the selected schedule type and scenario run mode. For detailed information about the schedule actions, see "Understanding Schedule Actions" on page 135.

You can modify these actions and, for real-life schedules, you can add other actions. For details, see "Managing Schedule Actions" on page 146.

Note: Schedule by Group only. You can copy schedule settings from one Vuser group to another. For more information, see "Copying Group Schedule Settings" on page 145.

---

a Start Group (Schedule by Group only). Set the group's schedule to start:

➤ Immediately upon starting the scenario run

➤ After the specified amount of time has elapsed

➤ After a specified group has finished running

b Initialize. Set the Vusers that will run in the scenario to initialize before they run as follows:

➤ Initialize all the Vusers at the same time

➤ Initialize a specified number of Vusers at given time intervals

➤ Initialize each Vuser just before it starts running

Note: Schedule by Group only. You can set all the Vusers in all the enabled Vuser groups to initialize before any of them start running. For more information, see "Initializing All Vuser Groups Before a Run" on page 143.

---

c Start Vusers/Duration/Stop Vusers. Specify how many Vusers to start running, how long they should run, and how many to stop running at a time. For more information about these actions, see "Understanding Schedule Actions" on page 135.

Optional Scenario Schedule Settings

This section describes additional options settings that you can apply to your scenario schedule.

This section includes:

➤ "Initializing All Vuser Groups Before a Run" below

➤ "Scheduling a Scenario Start Time" on page 144

Initializing All Vuser Groups Before a Run

To initialize all the Vusers in all the groups before running them, in the Schedule Definition area click the **Wait for all groups to finish initialization** button.

If the Initialize action of any of the Vuser groups in the scenario is set to **Initialize each Vuser just before it runs**, a message prompts you to change the Initialize action to **Initialize all Vusers simultaneously**. Click **Yes** to change the Initialize action setting.

Scheduling a Scenario Start Time

For both manual and goal-oriented scenarios, you specify when, after the Start Scenario command is issued, the scenario should actually begin running. You can set the scenario to start running without delay or after a specified amount of time. For details on starting a scenario, see Chapter 15, "Running a Scenario."

Note: When the scenario starts running, the selected scenario schedule starts running.

To schedule the start time of the scenario:

 1 In the Schedule Definition area, click the **Start Time** button. The Scenario Start Time dialog box opens.
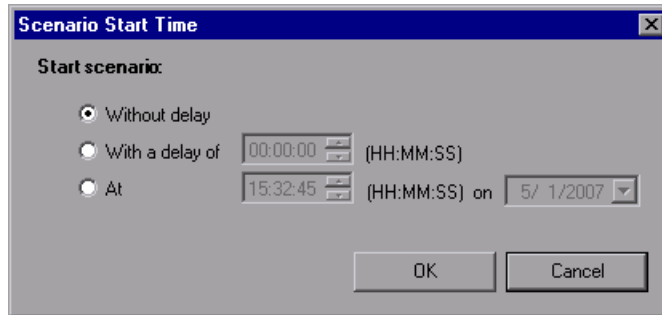
 2 Specify when, after the Start Scenario command has been issued, to start running the scenario. Select one of the following options:

➤ **Without delay.** Starts running the scenario immediately. This option is selected by default.

➤ **With a delay of** <00:00:00> **(HH:MM:SS).** Starts running the scenario after the specified time interval (in hours, minutes, and seconds) has passed.

➤ **At** <00:00:00> **(HH:MM:SS) on** <date>**.** Starts running the scenario on the specified date, at the specified time.

 3 Click **OK** to save your settings.

Copying Group Schedule Settings

If you are creating a schedule by Vuser group, and you have already defined schedule settings for one of the groups, you can copy those settings to other groups.

Note: Schedule settings include the schedule run mode and the set of actions.

To copy a Vuser group's schedule settings:

1 In the Scenario Groups pane, select a Vuser group.

2 On the Actions grid toolbar, click the **Copy Schedule Settings From** button. The Select Group dialog box opens.

3 Select the Vuser group from which to copy the schedule settings, and click **OK**.

The schedule settings are copied to the selected Vuser group.

Modifying Existing Schedules

You can modify a schedule by renaming it, changing its type or run mode, or adjusting the schedule actions.

In the **Schedule Name** box, select the schedule whose properties you want to modify, and follow the instructions described in "Creating Scenario Schedules" on page 140.

Note: When you switch from one run mode to another, all existing actions for the schedule are replaced by the default set of actions for the new run mode.

Deleting Schedules

To delete a schedule, in the **Schedule Name** box, select the schedule that you want to delete, and click the **Delete Schedule** button. The schedule is removed from the list.

Managing Schedule Actions

You add, edit, and delete schedule actions in the Actions grid.

Note: You can also modify schedule actions in the Interactive Schedule graph. For more information, see "Managing Schedules From the Interactive Schedule Graph" on page 158.

This section includes:

➤ "Schedule Actions Grid Toolbar" on page 146
➤ "Adding Schedule Actions" on page 148
➤ "Editing Schedule Actions" on page 150
➤ "Deleting Schedule Actions" on page 151

Schedule Actions Grid Toolbar

Toolbar Button	Button Name	Enables you to
	Add Action After	Add an action after the selected action
	Edit Action	Edit the selected action
	Delete Action	Delete the selected action
	Move Action Up	Move the selected action up in the grid

Toolbar Button	Button Name	Enables you to
⬇	**Move Action Down**	Move the selected action down in the grid
🔳	**Copy Schedule Settings From**	Copy another group's schedule settings to the selected group For details, see "Copying Group Schedule Settings" on page 145.
Total: 20 Vusers	**Total: XX Vusers**	View the number of Vusers in the scenario. **Note:** Editable in Percentage mode only.
↵	**Apply** (Visible in Percentage mode only, when modifying the total number of Vusers)	Apply the specified total number of Vusers proportionately to the Vuser groups. (Percentages are visible in the **%** column of the Scenario Scripts pane.)

Adding Schedule Actions

When you create a scenario, the Actions grid of the Scheduler lists default schedule actions. These actions are also displayed in the interactive schedule graph. In a real-life schedule, you can add actions to simulate a more true-to-life schedule by which to run your scenario.

Note: You can also add schedule actions in the Interactive Schedule graph. For more information, see "Managing Schedules From the Interactive Schedule Graph" on page 158.

To add an action to a schedule:

1 In the Actions grid do one of the following:

➤ Double-click the last row—marked by an asterisk (*****).

➤ Select an action after which to insert the new action, and click the **Add Action After** button.

Note: A new action is always added **after** the action selected in the Actions grid.

The Add Action dialog box opens.

2 In the **Action type** box, select the type of action to add.

Note: You can add a Start Vusers, Stop Vusers, or Duration action. For details about each action type, see "Understanding Schedule Actions" on page 135.

➤ If you are adding a Start Vusers or Stop Vusers action:

In the **Start/Stop** XX **Vusers** box, enter the number of Vusers to start/stop running, and select whether to:

➤ start/stop running all the Vusers simultaneously

➤ start/stop running the Vusers gradually

In this case, enter the number of Vusers to start/stop at a time, and at what time interval.

➤ If you are adding a Duration action, select how long the Scheduler should continue the preceding action before moving on to the next action.

3 Click **Apply**. The action is added to the Actions grid, and is displayed in the schedule graph.

At the same time, the Add Action dialog box displays the new action in Edit mode, in case you want edit the new action. If you edit the settings, click **Apply** again.

4 To add another action, click **Add Another Action** and repeat steps 2 and 3.

5 When you have finished adding actions to the schedule, click **OK** to close the Add Actions dialog box.

Editing Schedule Actions

You can edit the schedule actions in the Action grid.

Note: You can also edit schedule actions in the Interactive Schedule graph. For more information, see "Managing Schedules From the Interactive Schedule Graph" on page 158.

To edit a schedule action:

1 In the Actions grid, select the action that you want to edit.

Note: You can edit all types of actions. For details about each action type, see "Understanding Schedule Actions" on page 135.

---

2 Click the **Edit Action** button or double-click the action. The Edit Action dialog box opens.

Note: Alternatively, double-clicking the line in the Schedule graph opens the Edit Action dialog box.

---

    a Edit the action.

    b Click **Apply**. The details of the action are updated in the table.

    c If you want to edit another action, click **Next** or **Previous** to locate the action, and repeat steps a and b.

    d When you have finished editing the actions, click **OK**.

3 To move a selected action up in the Action grid, click the **Move Action Up** button.

4 To move a selected action down in the Action grid, click the **Move Action Down** button.

Deleting Schedule Actions

You delete schedule actions in the Actions grid.

Note: You can also delete schedule actions from the Interactive Schedule graph. For more information, see "Managing Schedules From the Interactive Schedule Graph" on page 158.
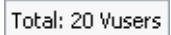

To delete a schedule action:

1 In the Actions grid, select the action you want to delete.

Note: You cannot delete Start Group or Initialize actions.

---

2 Click the **Delete Action** button, and click **Yes**.

The selected action is deleted from the schedule.

Viewing Scenario Schedules in the Interactive Graph

The Interactive Schedule graph provides a graphical representation of the scenario's schedule.
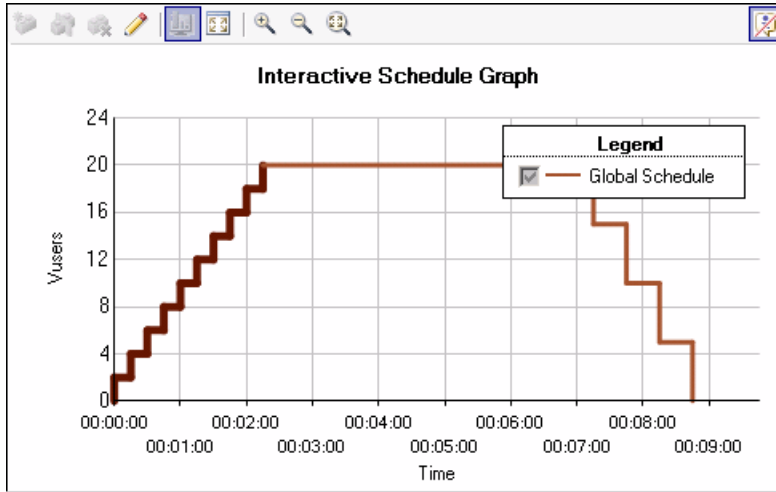
For real-life schedules, the graph is interactive, enabling you to modify the schedule from the graph itself. That is, you can create, edit, and delete schedule actions from the graph.

You can also watch the progress of a running schedule during a scenario run. You can pause the schedule during the run to modify settings, and resume running the schedule when you are ready.

This section includes:

➤ "Understanding the Interactive Schedule Graph Toolbar" on page 153

➤ "Viewing Vuser Group Schedules in the Interactive Graph" on page 155

➤ "Working with the Interactive Schedule Graph" on page 156

Understanding the Interactive Schedule Graph Toolbar

Toolbar Button	Button Name	Enables you to
	New Action	Append a new action to the last action in the graph. Available when the graph is in Edit mode only.
	Split Action	Split a selected action in two. Available when the graph is in Edit mode only.
	Delete Action	Delete a selected action from the schedule. Available when the graph is in Edit mode only.
	Edit/View Mode	Switch the graph display between Edit mode and View mode.
	Pause	Pause the schedule during the scenario run. Available only when the scenario is running.
	Resume	Resume running a schedule that was paused. Available only when the scenario is running.
	Show Selected Group	Display only the group selected in the Scenario Groups pane. Available for group schedules only.
	Show All Groups	Display all Vuser groups enabled in the scenario. Available for group schedules only.
	Open Full View	Open the graph in its own window.

Toolbar Button	Button Name	Enables you to
	Zoom In	Zoom into the x-axis of the graph— that is, spread out the graph to view shorter time intervals.
	Zoom Out	Zoom out of the x-axis of the graph— that is, view longer time intervals.
	Zoom Reset	Revert to the default time intervals displayed on the x-axis.
	Hide Legend	Hide the graph legend.
	Show Legend	Show the graph legend.

Viewing Vuser Group Schedules in the Interactive Graph

For group schedules, by default, all the Vuser groups' schedules are displayed in the interactive graph. You can also view a selected Vuser group's schedule on its own.

To view a particular Vuser group's schedule:

1 In the Scenario Groups pane, select the Vuser group. In the graph legend, the selected Vuser group becomes the default Vuser group in the graph. The option to clear the check box becomes unavailable.
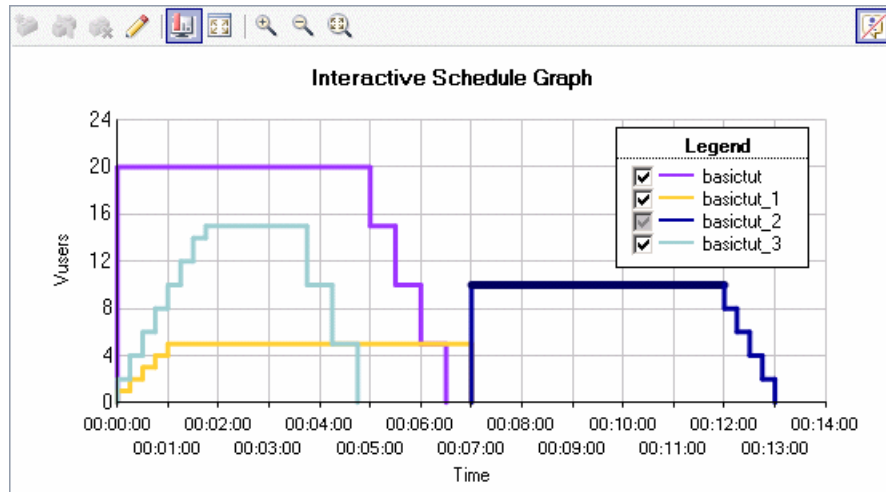
2 Click the **Show Selected Group** button. Only the selected Vuser group is displayed in the graph.
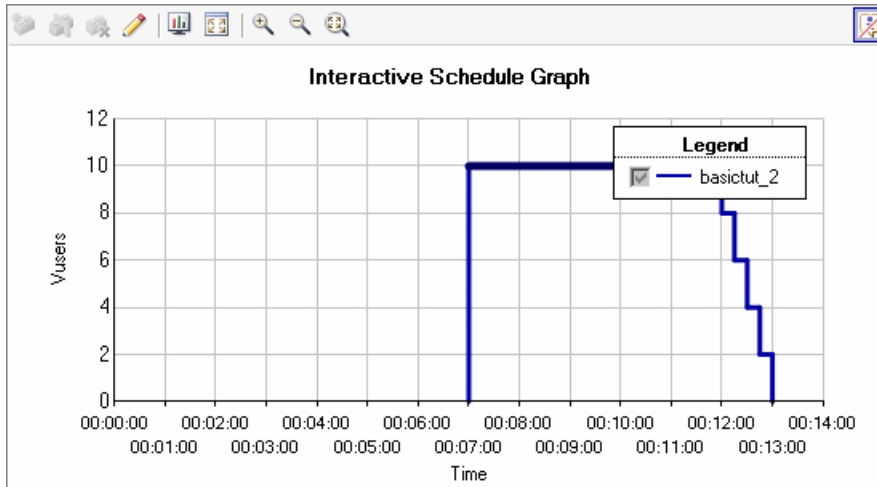
To view all of Vuser groups again, click the **Show All Groups** button.

Working with the Interactive Schedule Graph

You can view an enlarged version of the graph, zoom in and out of the time line of the graph, and view and hide the graph legend.

Opening the Interactive Schedule Graph in Full View

You can view an enlarged version of the schedule's interactive graph by opening the graph in its own window.

To open the graph in full view:

➤ In the interactive graph pane, click the **Open Full View** button.

A window opens displaying the graph pane in full view.

Note: All the options available in the interactive graph pane are also available in the full-view graph window.

Zooming Into and Out of the Timeline of the Graph

You can spread out the timeline (x-axis) of the schedule graph, and view the schedule details over shorter time intervals.

➤ To spread out the timeline, click the **Zoom in** button.

Depending on the duration of the schedule, the graph might spread wider than the graph pane. In this case, a **Time** scroll bar appears. Use the scroll bar to scroll across the graph.
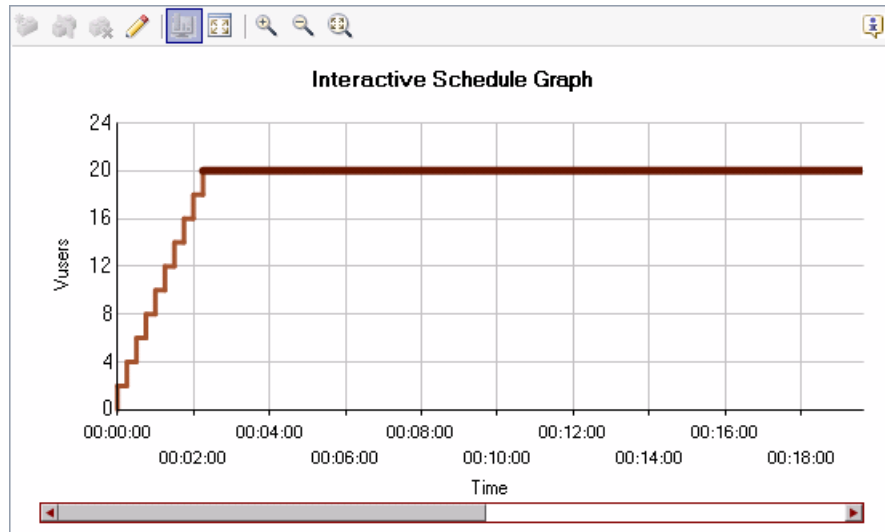
➤ To zoom out of the timeline, click the **Zoom out** button.

➤ To revert to the default time intervals displayed on the x-axis, click the **Zoom Reset** button.

Showing and Hiding the Graph Legend

➤ To hide the legend of the interactive schedule graph, click the **Hide Legend** button in the upper right corner of the graph pane or full-view window.

➤ To show the legend of the interactive schedule graph, click the **Show Legend** button in the upper right corner of the graph pane or full-view window.

157

Managing Schedules From the Interactive Schedule Graph

The Schedule graph is interactive, enabling you to edit the scenario schedule from the graph itself. Each line in the graph represents an action in the Actions grid. Only the Start Vusers, Duration, and Stop Vusers actions are displayed in the graph. When you modify these actions in the graph, the action is updated in the Actions grid.

Note: Classic schedules can only be viewed in the interactive schedule graph. To modify a classic schedule, you must edit the actions in the Actions grid.

This section includes:

➤ Switching Between Edit and View Mode

➤ Adding an Action

➤ Editing an Action

➤ Deleting an Action

Switching Between Edit and View Mode

By default, the Schedule graph is displayed in View mode. In order to edit the graph, you must switch the graph to Edit mode.

To switch the between Edit and View modes, click the **Edit/View Mode** button.

Adding an Action

You add schedule actions in the graph by splitting an action into two actions, or appending a new action after the last action in the graph.

To add an action by splitting one action into two actions:

1 Ensure that the graph is in Edit mode (click the **Edit Mode** button).

2 Select the line that represents the action that you want to split.

Tip: Selecting the action in the Actions grid highlights the corresponding line in the graph.

3 Click the **Split Action** button. The selected line is split in two. The original action in the Actions grid is split into two equivalent actions, each representing half of the original action.

For example,

➤ splitting a Duration action of 5 minutes results in two Duration actions of 2.5 minutes each

➤ splitting a Start Vusers action that starts 20 Vusers results in two Start Vusers actions, each starting 10 Vusers

To append an action after the last action:

1 Ensure that the graph is in Edit mode (click the **Edit Mode** button).

2 In the graph pane toolbar, click the **New Action** button.

3 Append a new actions as follows:

➤ To append a Start Vusers action, click the graph anywhere to the right of and above the last line of the graph

➤ To append a Duration action, click the graph anywhere to the right of the last line of the graph

➤ To append a Stop Vusers action, click the graph anywhere to the right of and below the last line of the graph

4 To edit the action, see "Editing an Action" below, or see "Editing Schedule Actions" on page 150.

Editing an Action

You can edit a schedule action in the graph by dragging the endpoint of the line representing it from one location to another.

To edit a schedule action:

1 Ensure that the graph is in Edit mode (click the **Edit Mode** button).

2 In the Actions grid, select the action you want to edit, or click the corresponding line in the graph. The line is highlighted and its endpoints, a dot and a diamond, are displayed.

3 Modify the action as described in the following table. The action details displayed in the Actions grid are updated accordingly.

Note: Alternatively, you can double-click the line in the graph to open the Edit Action dialog box, and modify the action as described in "Editing Schedule Actions" on page 150.

Action	Modification
Start Vusers	➤ To increase the number of Vusers to start running, drag the diamond-shaped endpoint upwards. ➤ To decrease the number of Vusers to start running drag the diamond-shaped endpoint downwards.
Start Vusers	➤ To increase the time interval between starting Vusers, drag the diamond-shaped endpoint to the right. ➤ To decrease the time interval between starting Vusers, drag the diamond-shaped endpoint to the left. **Note:** A vertical line indicates that the Vusers start running simultaneously.
Duration	➤ To increase the time interval between scheduled actions, drag the diamond-shaped endpoint to the right. ➤ To decrease the time interval between scheduled actions, drag the diamond-shaped endpoint to the left.

Action	Modification
Stop Vusers	➤ To decrease the number of Vusers to stop running, drag the diamond-shaped endpoint upwards. ➤ To increase the number of Vusers to stop running, drag the diamond-shaped endpoint downwards.
Stop Vusers	➤ To increase the time interval between stopping Vusers, drag the diamond-shaped endpoint to the right. ➤ To decrease the time interval between stopping Vusers, drag the diamond-shaped endpoint to the left. **Note:** A vertical line indicates that the Vusers stop running simultaneously.

Tip: To fine-tune any of the details of the selected action line, use the arrow keys on your keyboard.

Deleting an Action

You can delete a schedule action from the schedule graph.

To delete a schedule action:

1 Ensure that the graph is in Edit mode (click the **Edit Mode** button).

2 In the Actions grid or in the graph, select the action you want to delete.

3 In the graph pane toolbar, click the **Delete Action** button, and click **Yes**.

The selected action is deleted from the graph.

Viewing the Schedule Progress During a Run

While the scenario is running, you can view the progress of the scenario schedule in the schedule graph pane. The progress is indicated by a vertical red line that slides across the graph during the run.

Important: The scenario schedule may run a few seconds ahead of the scenario run.
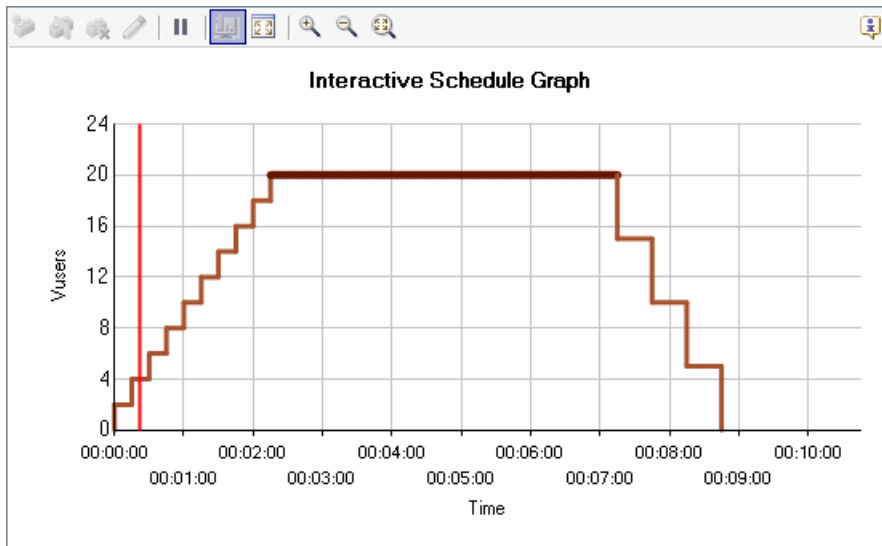
At any point during the run, you can pause the schedule, and resume running the schedule.

➤ To pause the schedule, in the graph pane or full-view graph window, click the **Pause** button.

➤ To resume the schedule, in the graph pane or full-view window click the **Resume** button.

Adding Vusers to a Scheduled Scenario During a Run

When you run a scenario or Vuser group, the scenario schedule settings are applied to all Vusers that are manually added to the scenario or Vuser group during the scenario run. For example, if a running scenario or Vuser group has a set duration of five minutes, all Vusers subsequently added to the scenario or Vuser group run only for the remaining part of this time period.

Vusers added to a scenario or Vuser group which has finished running are not affected by schedule settings and run according to the scenario run-time settings.

For more information on controlling Vusers manually , see "Manually Adding Vusers to a Running Scenario" on page 278.

8

Defining Service Level Agreements

Service level agreements (SLAs) enable you to define goals for your load test scenario. During a scenario run, the Controller measures the performance and collects data. Analysis compares this data against thresholds defined in the SLAs.

This chapter includes:

About Defining Service Level Agreements

When you design a load test scenario, you can define goals or **service level agreements** (SLAs) for the performance metrics. When you run the scenario, LoadRunner gathers and stores performance-related data. When you analyze the run, Analysis compares this data against the SLAs and determines SLA statuses for the defined measurements.

Depending on the measurements that you are evaluating, LoadRunner determines SLA statuses in one of the following ways:

➤ **SLA status determined per time interval within the run.** Analysis displays SLA statuses at set time intervals—for example, every 10 seconds—over a timeline within the run. For more information, see "Defining an SLA Goal Measured Per Time Interval" on page 167.

➤ **SLA status determined over the whole run.** Analysis displays a single SLA status for the whole scenario run. For more information, see "Defining an SLA Goal Measured Over the Whole Run" on page 175.

Note: You can define and edit SLAs in the Controller or in Analysis.

For details about viewing SLA information in Analysis reports, refer to the *HP LoadRunner Analysis User's Guide*.

Defining an SLA Goal Measured Per Time Interval

For the **Average Transaction Response Time** and **Errors per Second** measurements, Analysis displays SLA statuses at set time intervals over a timeline within the run.

That is, at each time interval in the timeline—for example, every 10 seconds—Analysis checks to see if the measurement's performance has deviated from the threshold defined in the SLA.

When creating SLAs for these measurements, you can also specify load criteria to take into account when comparing the data.

➤ To define SLAs for Average Transaction Response Time, see "SLAs for Average Transaction Response Time" below.

➤ To define SLAs for Errors per Second, see "SLAs for Errors Per Second" on page 171.

➤ To set the time intervals which Analysis uses for evaluating the SLA, see "Selecting a Tracking Period" on page 174.

SLAs for Average Transaction Response Time

This section explains how to define an SLA for evaluating the status of Average Transaction Response Time during the run. For Average Transaction Response Time, LoadRunner evaluates SLA statuses at set time intervals within the run.

Note: To set the time intervals that Analysis uses for evaluating the SLA, see "Selecting a Tracking Period" on page 174.

To define an SLA for Average Transaction Response Time:

1 Open the Service Level Agreement Wizard.

➤ **If you are working in Analysis**: Select **Tools** > **Configure SLA Rules** to open the Service Level Agreement dialog box. Click **New**.

➤ **If you are working in the Controller**: In the Design tab, in the Service Level Agreement pane, click **New**.

The Service Level Agreement wizard opens.

2 Complete the SLA Wizard steps, as described in the following sections.

Measurement—Select a Measurement for Your Goal

For Average Transaction Response Time, LoadRunner evaluates SLA statuses at set time intervals within the run.

Under **SLA status determined per time intervals over a timeline**, select **Average Transaction Response Time**.

Transactions—Select Transactions

From the **Available Transactions** list, select the transactions that you want to evaluate as part of your SLA and click **Add**.

Note: You can select multiple transactions using the CTRL key.

The transactions you selected are displayed in the **Selected Transactions** list.

Load Criteria—Set Load Criteria

Select load criteria for your goal and define appropriate load value ranges.

Note: In the next wizard step (Thresholds page), you will set different thresholds per each of these load value ranges.

➤ In the **Load Criteria** box, select the relevant load criteria that you want to use—for example, **Running Vusers**.

To define an SLA without load criteria, select **None** in the **Load Criteria** box.

➤ In the **Less than** box, set the lower load value range by entering a maximum value for this range. This range is between 0 and the maximum value you entered, but does not include the maximum value.

➤ To set in-between load value ranges, select **Between** from the **Greater than or equal to/Between** combo box and enter minimum and maximum values for the range. The minimum value is included in the range and the maximum value is not.

Note: You can set up to three in-between ranges.

➤ To set the upper load value range, select **Greater than or equal to** from the **Greater than or equal to/Between** combo box and enter the minimum value for this range. The minimum value is included in this range.

Note: Valid load value ranges are consecutive—there are no gaps in the range—and span all values from zero to infinity.

Thresholds—Set Threshold Values

Set maximum thresholds for each transaction that you are evaluating.

➤ If you defined load criteria in the previous step, you set thresholds for each transaction per the defined load value ranges.

➤ If you did not define load criteria, you set a single threshold for each transaction.

Enter the relevant thresholds (per load criteria, if defined) in the table displayed at the top the page.

	Running Vusers		
Transaction Name	<5	≥5 and <10	≥10
Action_Transaction	0	0	0
vuser_end_Transact...	0	0	0

Note: If Average Transaction Response Time exceeds the maximum threshold value during a particular time interval during the run, LoadRunner Analysis displays an SLA status of **Failed** for that time interval.

If you want to apply one set of threshold values to all transactions, enter the threshold values in the table displayed at the bottom of the page and click **Apply to all transactions**. These values are applied to all the transactions in the table at the top of the page.

Tip: After you apply one set of threshold values to all transactions, you can manually modify the thresholds of individual transactions in the table at the top of the page.

Finish

If you want to define another SLA after you save the current one, select **Define another SLA.**

Click **Finish** to save the SLA.

Note: For details about viewing SLA information in Analysis reports, refer to the *HP LoadRunner Analysis User's Guide*.

SLAs for Errors Per Second

This section explains how to define an SLA for evaluating the status of Errors per Second during the run. For Errors per Second, LoadRunner evaluates SLA statuses at set time intervals within the run.

Note: To set the time intervals that Analysis uses for evaluating the SLA, see "Selecting a Tracking Period" on page 174.

To define an SLA for Errors per Second:

1 Open the Service Level Agreement Wizard.

   ➤ **If you are working in Analysis**: Select **Tools** > **Configure SLA Rules** to open the Service Level Agreement dialog box. Click **New.**

   ➤ **If you are working in the Controller**: In the Design tab, in the Service Level Agreement pane, click **New.**

   The Service Level Agreement wizard opens.

2 Complete the SLA Wizard steps, as described in the following sections.

Measurement—Select a Measurement for Your Goal

For Errors per Second, LoadRunner evaluates SLA statuses at set time intervals within the run.

Under **SLA status determined per time interval within the run**, select **Errors per Second**.

Load Criteria—Set Load Criteria

Select load criteria for your goal and define appropriate load value ranges.

Note: In the next wizard step (Thresholds page), you will set different thresholds per each of these load value ranges.

➤ In the **Load Criteria** box, select the relevant load criteria that you want to use—for example, **Running Vusers**.

To define an SLA without load criteria, select **None** in the **Load Criteria** box.

➤ In the **Less than** box, set the lower load value range by entering a maximum value for this range. This range is between 0 and the maximum value you entered, but does not include the maximum value.

➤ To set in-between load value ranges, select **Between** from the **Greater than or equal to/Between** combo box and enter minimum and maximum values for the range. The minimum value is included in the range and the maximum value is not.

Note: You can set up to three in-between ranges.

➤ To set the upper load value range, select **Greater than or equal to** from the **Greater than or equal to/Between** combo box and enter the minimum value for this range. The minimum value is included in this range.

Note: Valid load value ranges are consecutive—there are no gaps in the range—and span all values from zero to infinity.

Thresholds—Set Threshold Values

In the Thresholds page you set a maximum threshold value for Errors per Second.

➤ If you defined load criteria in the previous step, enter threshold values per the defined load value ranges, in the table provided.

➤ If you did not define load criteria, enter the maximum threshold value in the **Threshold** box.

Note: If Errors per Second exceeded the maximum threshold value during a particular time interval during the run, LoadRunner Analysis displays an SLA status of **Failed** for that time interval.

Finish

If you want to define another SLA after you save the current one, select **Define another SLA.**

Click **Finish** to save the SLA.

Note: For details about viewing SLA information in Analysis reports, refer to the *HP LoadRunner Analysis User's Guide*.

Selecting a Tracking Period

When you define a service level agreement for measurements that are evaluated over a timeline, LoadRunner determines SLA statuses at specified time intervals within that timeline. These time intervals are known as the **tracking period** of the load test scenario.

To define the tracking period for the scenario:

 1 In the Design tab, in the Service Level Agreement pane, click the **Advanced** button. The Advanced Options window opens.

 2 Select one of the following methods to determine the tracking period for the scenario.

 ➤ **Internally calculated tracking period.**

   Analysis sets the tracking period to the minimum value possible, taking into account the aggregation granularity defined for the scenario. This value is at least 5 seconds. It uses the following formula:

   ```
   Tracking Period = Max (5 seconds, aggregation granularity)
   ```

 ➤ **Tracking period of at least X seconds.**

   Select a value (X) to determine the minimum amount of time for the tracking period. This value can never be less than 5 seconds.

Analysis sets the tracking period to the nearest multiple of the scenario's aggregation granularity that is greater than or equal to the value (X) that you selected.

Assume for example that you select a tracking period of X=10, and assume that the aggregation granularity for the scenario is 6. The tracking period is set to the nearest multiple of 6 that is greater than or equal to 10, that is, Tracking Period = 12.

For this option, Analysis uses the following formula:

Tracking Period = Max(5 seconds, m(Aggregation Granularity))

where m is a multiple of the scenario's aggregation granularity such that
m(Aggregation Granularity) ≥ X

3 Click **OK**.

Defining an SLA Goal Measured Over the Whole Run

For certain measurements—Total Hits, Average Hits, Total Throughput, and Average Throughput—Analysis displays a single SLA status for the whole load test scenario run. This section explains how to define an SLA for these measurements.

To define an SLA that evaluates the status over a whole run:

1 Open the Service Level Agreement Wizard.

➤ **If you are working in Analysis**: Select **Tools** > **Configure SLA Rules** to open the Service Level Agreement dialog box. Click **New**.

➤ **If you are working in the Controller**: In the Design tab, in the Service Level Agreement pane, click **New**.

The Service Level Agreement wizard opens.

2 Complete the SLA Wizard steps, as described in the following sections.

Measurement—Select a Measurement for Your Goal

For the following measurements, Analysis displays a single SLA status for the whole scenario run:

➤ Total Hits

➤ Average Hits (Hits per second)

➤ Total Throughput

➤ Average Throughput (Bytes per second)

Under **SLA status determined over the whole run**, select the relevant measurement.

Thresholds—Set Threshold Values

The measurement that you selected in the previous step is displayed in the **Measurement** box.

In the **Threshold** box, enter the minimum threshold value for your measurement.

Note: If the measurement value fell below this minimum threshold value during the run, LoadRunner Analysis displays an SLA status of **Failed** for the entire run.

Finish

If you want to define another SLA after you save the current one, select **Define another SLA.**

Click **Finish** to save the SLA.

Note: For details about viewing SLA information in Analysis reports, refer to the *HP LoadRunner Analysis User's Guide.*

Understanding the Service Level Agreement Pane

The Service Level Agreement pane includes the following options:

Function	Button	Enables You To
New		Define a new SLA. See: ➤ "Defining an SLA Goal Measured Per Time Interval" on page 167 ➤ "Defining an SLA Goal Measured Over the Whole Run" on page 175
Details		View the details of the SLA selected in the Service Level Agreement pane. The Service Level Agreement - details dialog box opens, displaying a summary of the details for the selected SLA.
Edit		Edit the definitions of the SLA selected in the Service Level Agreement pane. The SLA wizard opens with the details of the selected SLA.
Delete		Delete the SLA selected in the Service Level Agreement pane.
Advanced		Adjust the tracking period for SLAs that are evaluated per time interval over a timeline. See "Selecting a Tracking Period" on page 174.

9

Creating a Goal-Oriented Scenario

You build a goal-oriented scenario for an application by defining the goals you want your test to achieve. This chapter describes how to create a goal-oriented scenario.

This chapter includes:

About Planning a Goal-Oriented Scenario

In a goal-oriented scenario, you define the goals you want your test to achieve, and LoadRunner automatically builds a scenario for you based on these goals. You can define five types of goals in a goal-oriented scenario: the number of virtual users, the number of hits per second (Web Vusers only), the number of transactions per second, the number of pages per minute (Web Vusers only), or the transaction response time you want your scenario to reach. You define one of these scenario goals using the Edit Scenario Goal dialog box. For more information on this dialog box, see "Defining Scenario Goals" on page 184.

Note: To run a Transactions per Second or Transaction Response Time goal type, your script must contain transactions. For each of these goal types, you define the transaction in your script that you want to test.

Virtual Users Goal Type

If you want to test how many Vusers your application can run simultaneously, it is recommended that you define a Virtual Users goal type. Running this type of goal-oriented scenario is similar to running a manual scenario. For more information on defining this goal type, see "Defining Scenario Goals" on page 184.

Pages per Minute and Hits/Transactions per Second Goal Types

If you want to test the strength of your server, it is recommended that you define a Hits per Second, Pages per Minute, or Transactions per Second goal type. Specify a minimum-maximum range of Vusers for LoadRunner to run, and a Transaction Name for the Transactions per Second goal type.

The Controller attempts to reach the goal you defined using a minimum number of Vusers. If it cannot reach this goal using the minimum number of Vusers, the Controller increases the number of Vusers until the maximum number you defined is reached. If your goal cannot be reached with the maximum number of Vusers you specified, increase this number and execute your scenario again.

For more information regarding the formula used by the Controller in running the Pages per Minute and Hits/Transactions per Second goal types, see "Understanding the Hits/Transactions per Second and Pages per Minute Goal Types" on page 189.

Transaction Response Time Goal Type

If you want to test how many Vusers can be run simultaneously without exceeding a desired transaction response time, it is recommended that you define a Transaction Response Time goal type. Specify the name of the transaction in your script that you want to test, and a minimum-maximum range of Vusers for LoadRunner to run. The transaction response time you specify should be a pre-defined threshold value. For example, if you do not want a customer to wait more than five seconds to log in to your e-commerce site, specify a maximum acceptable transaction response time of five seconds. Set the minimum and maximum number of Vusers to the minimum-maximum range of customers you want to be able to serve simultaneously.

If the scenario does not reach the maximum transaction response time that you defined, your server is capable of responding within a reasonable period of time to the number of customers you want to be able to serve simultaneously. If the defined response time is reached after only a portion of the Vusers has been executed, or if you receive a message that the defined response time will be exceeded if the Controller uses the maximum number of Vusers defined, you should consider revamping your application and/or upgrading your server software and hardware.

> **Note:** In order for a Transaction Response Time goal-oriented scenario to be effective, you must choose your transaction carefully, ensuring that it performs effective hits on the server.

Understanding the Goal-Oriented Scenario Design Tab

When you create a goal-oriented scenario, the Controller displays the Scenario Scripts, Scenario Goal, and Service Level Agreement panes in the Design tab.

The **Scenario Scripts** pane lists all enabled and disabled Vuser scripts, their paths, the percentage of the total target assigned to each script, and the load generators. For more information on the Scenario Scripts pane, see "Assigning Properties to Scripts" on page 191.

The **Scenario Goal** pane displays information regarding the goal profile: its name, the goal you define, a minimum and maximum number of Vusers, the duration of the scenario, and the load behavior.

You can define five types of goals in a goal-oriented scenario: the number of virtual users, the number of hits per second (Web Vusers only), the number of transactions per second, the number of pages per minute (Web Vusers only), or the transaction response time you want your scenario to reach. For more information on defining goal types, see "Defining Scenario Goals" on page 184.

The **Service Level Agreement** pane lists the service level agreements defined for the scenario. For more information, see Chapter 8, "Defining Service Level Agreements."

You can perform the following actions on a goal profile or scenario:

➤ Define the goal profile name and goal type

➤ Add a new script to the scenario and configure it

➤ Add one or more load generators to a script and configure the load generators

➤ Enable or disable a script for the scenario

➤ Define the duration of the scenario and ramp-up behavior

➤ Define service level agreements for the scenario

➤ Run the scenario

➤ Stop the scenario

➤ Reset the scenario

➤ Configure scenario result settings

Defining Scenario Goals

You define scenario goal settings for a goal-oriented scenario from the Edit Scenario Goal dialog box.

To define a scenario goal:

1 Click **Edit Scenario Goal** in the Scenario Goal pane or select **Scenario** > **Goal Definition**. The Edit Scenario Goal dialog box opens.
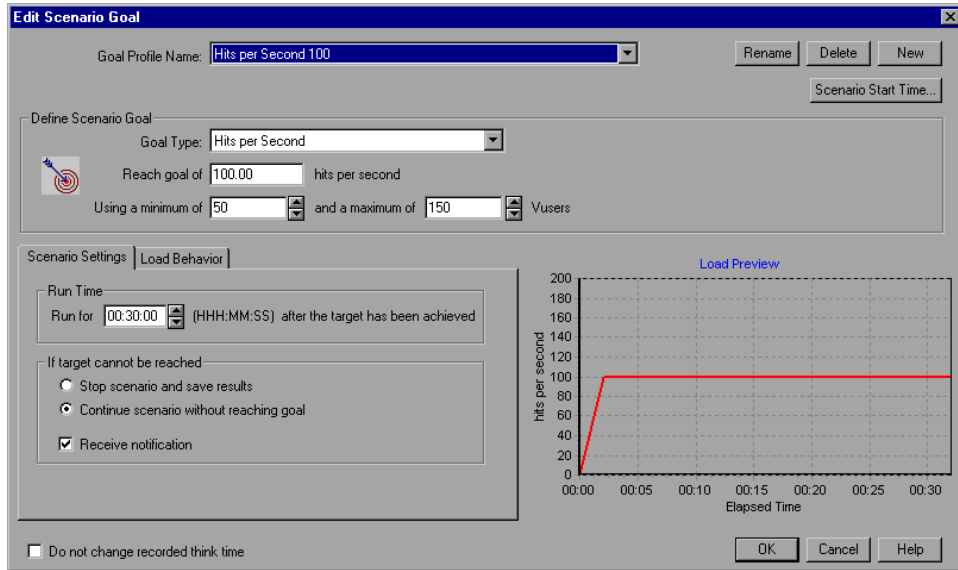
2 Select a **Goal Profile Name**. To enter a new name, click **New**, type the new goal profile name in the New Goal Profile dialog box, and click **OK**. The new goal profile name appears in the selector.

3 In the Define Scenario Goal box, select a **Goal Type** as described in "Understanding the Edit Scenario Goal Dialog Box" on page 185.

Note: VuGen automatically defines each **Init**, **Action**, and **End** unit as a transaction. In addition, you can insert a static transaction in your script using the Start Transaction and End Transaction functions.

---

4 In the **Scenario Settings** tab, specify how long you want to run your scenario after the target is achieved, and whether to continue the scenario if the target cannot be reached. For more information, see "Understanding the Scenario Settings Tab" on page 187.

5 Select the **Load Behavior** tab, and specify how and when you want the Controller to reach your target. For more information, see "Understanding the Load Behavior Tab" on page 188.

6 Select **Do not change recorded think time** if you want LoadRunner to run the scenario using the think time recorded in your script.

7 Click **OK** to close the Edit Scenario Goal dialog box. The scenario target information you entered appears in the Scenario Goal window.

Note: When you run a goal-oriented scenario, the goal you defined is displayed in the appropriate graph, along with the scenario results. This enables you to compare the results with your target goal.

Understanding the Edit Scenario Goal Dialog Box

The Edit Scenario Goal enables you to define scenario information for a goal-oriented scenario.

➤ **Goal Profile Name.** Select a goal profile name.

➤ **Rename.** Rename a goal profile using the New Goal Profile dialog box.

➤ **Delete.** Delete a goal profile from the list of goal profile names.

➤ **New.** Enter a new goal profile name using the New Goal Profile dialog box.

➤ **Scenario Start Time.** Opens the Scenario Start dialog box, enabling you to start running a goal-oriented scenario at a later point in time.

➤ **Define Scenario Goal.**

➤ **Goal Type.** Select the type of goal you want for your scenario.

➤ **Pages per Minute (Web Vusers only).** Enter a target number of downloaded pages per minute that you would like your scenario to reach, and select a minimum and maximum number of Vusers for the scenario.

➤ **Virtual Users.** Enter a target number of virtual users that you would like your scenario to reach.

➤ **Hits per Second (Web Vusers only).** Enter a target number of hits per second (HTTP requests per second) that you would like your scenario to reach, and select a minimum and maximum number of Vusers for the scenario.

➤ **Transactions per Second.** Enter a target number of transactions per second that you would like your scenario to reach, and select a minimum and maximum number of Vusers for the scenario. In addition, select a static script transaction for your scenario to test, or enter the name of an automatic script transaction that you have recorded in the Transaction Name box.

➤ **Transaction Response Time.** Enter a target transaction response time that you would like your scenario to reach, and select a minimum and maximum number of Vusers for the scenario. In addition, select a static script transaction for your scenario to test, or enter the name of a dynamic script transaction that you have recorded in the Transaction Name box.

➤ **Do not change recorded think time.** Instructs LoadRunner to run the scenario using the think time recorded in your script.

Note: If you select this option, you may need to increase the number of Vusers in your scenario in order to reach your target.

➤ **Load Preview.** Displays a graph of the goal and load behavior you defined.

Understanding the Scenario Settings Tab

The Scenario Settings Tab lets you specify how long you want your scenario to run after the target is achieved, and whether to continue the scenario if the target you defined cannot be reached.
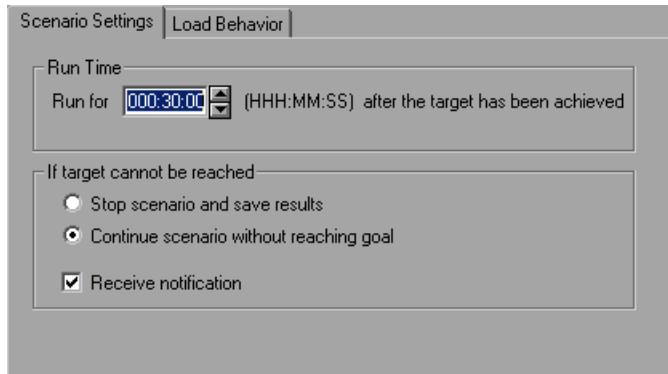
➤ **Run Time.**

   ➤ **Run for X (HH:MM:SS) after the target has been achieved.** Select the amount of time you want your scenario to run after reaching the target.

➤ **If target cannot be reached.** Select one of the following two options:

   ➤ **Stop scenario and save results.** Instructs the Controller to stop the scenario and save the scenario results, if the target you defined cannot be reached.

   ➤ **Continue scenario without reaching goal.** Instructs the Controller to continue running the scenario, even if the target you defined cannot be reached.

   ➤ **Receive notification.** Instructs the Controller to send you an error message indicating that the target cannot be reached.

187

Understanding the Load Behavior Tab

The Load Behavior tab lets you specify how and when you want the
Controller to reach your target.
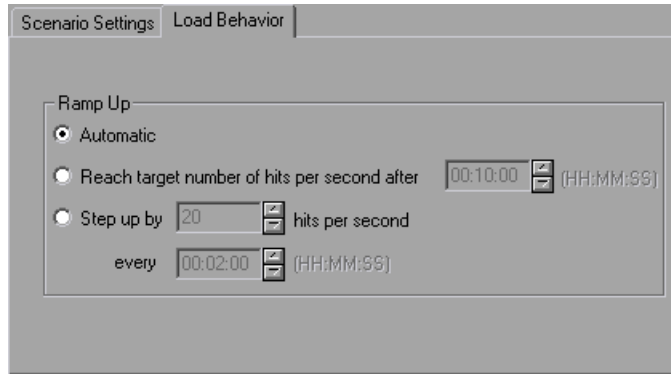
➤ **Ramp Up.** Select one of the following options:

  ➤ **Automatic.** Instructs the Controller to run the default number of Vusers
  in a batch (50 Vusers—or all the Vusers, if the maximum number of
  Vusers defined is less than 50—every two minutes).

  ➤ **Reach target X after.** Select the amount of scenario time you want to
  elapse before the Controller reaches your target.

  ➤ **Step up by** (not available for the Transactions per Second and
  Transaction Response Time goal types)**.** Select the gradation according to
  which you want the Controller to reach your target (x number of virtual
  users/hits/pages every x amount of time).

Understanding the Hits/Transactions per Second and Pages per Minute Goal Types

When you define a Pages per Minute or Hits/Transactions per Second goal type, the Controller divides the target you defined by the minimum number of Vusers you specified, and determines the target number of hits/transactions per second or pages per minute that each Vuser should reach. The Controller then begins loading the Vusers according to the load behavior settings you defined, as follows:

➤ If you selected to run the Vusers automatically, LoadRunner loads fifty Vusers in the first batch. If the maximum number of Vusers defined is less than fifty, LoadRunner loads all of the Vusers simultaneously.

➤ If you chose to reach your target after a certain period of the scenario elapses, LoadRunner attempts to reach the defined target within this period of time. It determines the size of the first batch of Vusers based on the time limit you defined and the calculated target number of hits, transactions, or pages per Vuser.

➤ If you chose to reach your target by gradation (x number of pages/hits every x amount of time), LoadRunner calculates the target number of hits or pages per Vuser and determines the size of the first batch of Vusers accordingly.

Note: The last load behavior option is not available for the Transactions per Second goal type.

After running each batch of Vusers, LoadRunner evaluates whether the target for the batch was achieved. If the batch target was not reached, LoadRunner recalculates the target number of hits, transactions, or pages per Vuser, and readjusts the number of Vusers for the next batch to be able to achieve the defined goal. By default, a new batch of Vusers is released every two minutes.

If the goal has not been reached once the Controller has launched the maximum number of Vusers, LoadRunner attempts to reach the defined target once more by recalculating the target number of hits, transactions, or pages per Vuser, and running the maximum number of Vusers simultaneously.

A Pages per Minute or Hits/Transactions per Second goal-oriented scenario is assigned a "Failed" status if:

➤ the Controller has twice attempted to reach the goal using the maximum number of Vusers specified, and the goal could not be reached.

➤ no pages per minute or hits/transactions per second were registered after the first batch of Vusers was run.

➤ the number of pages per minute or hits/transactions per second did not increase after the Controller ran a certain number of Vuser batches.

➤ all the Vusers that were run failed.

➤ there were no available load generators for the type of Vusers you attempted to run.

Assigning Properties to Scripts

The Scenario Scripts pane displays a list of scripts you selected for the scenario.
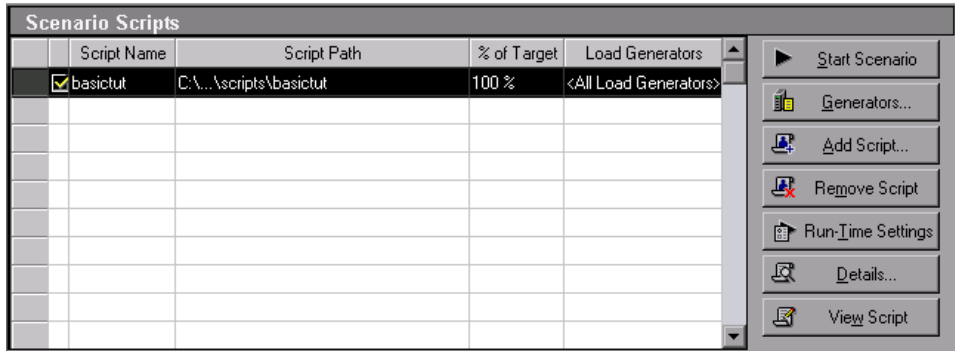
The **% of Target** column indicates the percentage of the overall target number of Vusers, pages per minute, hits per second, transactions per second, or transaction response time that is automatically distributed to each Vuser script. The **Load Generators** column automatically contains **<All Load Generators>** for each Vuser script.

To modify the percentage of Vusers assigned to a script:

In the script's **% of Target** column, enter a percentage of the total target number of Vusers, pages per second, hits per second, transactions per second, or transaction response time you want LoadRunner to reach during the scenario. During the scenario, LoadRunner attempts to reach the percentage of the total that you stipulate for each script in the scenario.

To add or modify load generators for a script:

 1 In the script's **Load Generators** column, select one or more load generators from the **Load Generator Name** list, and click **OK**. If you select multiple load generators, the Vusers assigned to the script are distributed evenly among the load generators.

 2 Alternatively, you can select **<Add..>** to add a load generator to the list. The Add New Load Generator dialog box opens
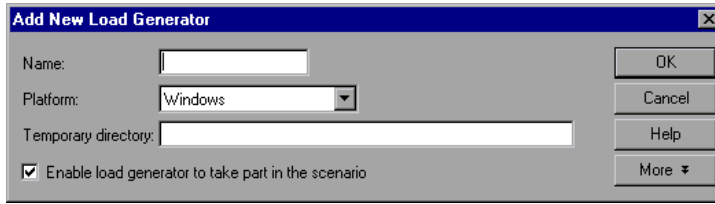
 3 In the **Name** box, type the name of the load generator.

 4 In the **Platform** box, select the type of platform on which the load generator is running.

 5 In the **Temporary Directory** box, type a location of the load generator where the Controller can store temporary files, or leave the field empty to accept the default location. By default, LoadRunner stores temporary files on the load generator during scenario execution, in a temporary directory specified by the load generator's TEMP or TMP environment variables.

 6 To allow the load generator to take part in the scenario, check **Enable load generator to take part in the scenario**.

 7 Click **More** to expand the dialog box and show the Add Load Generator tabs. For information on configuring settings for each load generator, see "Configuring Load Generator Settings" on page 88.

8 Click **OK** to close the Add Load Generator dialog box. LoadRunner adds the new load generator to the Load Generator Name list. To include the new load generator in your scenario, select it from the Load Generator Name list, and click **OK**.

Note: You can select multiple load generators.

Repeat the above procedure for each load generator you want to add to your scenario.

Configuring Load Generators

You can set a load generator's attributes while adding it to the load generator list, or modify the attributes of an existing load generator at any time, using the Load Generators dialog box. You can also use the Load Generators dialog box to indicate which load generators will run Vusers in the scenario. For example, if a load generator is unavailable for a particular scenario run, you can use the Load Generators dialog box to exclude it temporarily instead of removing it entirely from your list of load generators. For instructions on using the Load Generators dialog box, see "Configuring Load Generators" on page 84. To configure additional load generator settings, see "Configuring Load Generator Settings" on page 88.

To configure global settings for all load generators participating in the scenario, use LoadRunner's Options dialog box. For more information, see Chapter 11, "Configuring a Scenario."

Load Balancing

Load balancing evenly distributes the load generated by Vusers among the requested load generators, ensuring an accurate load test.

When a Windows load generator's CPU usage becomes overloaded, the Controller stops loading Vusers on the overloaded load generator, and automatically distributes them among load generators taking part in the scenario. Only where there are no other load generators in the scenario does the Controller stop loading Vusers.

You can monitor the status of a machine's CPU usage using the icons in the Load Generators dialog box. When the CPU usage of a load generator becomes problematic, the icon to the left of the load generator name contains a yellow bar. When the machine becomes overloaded, the icon contains a red bar.

Note: Load balancing is only available in goal-oriented scenarios and manually controlled scenarios in the Percentage Mode.

Configuring Scripts

You can add a script to the Scenario Scripts list using the Add Script dialog box. Once you have added a script to the list, you can view the details of the script you selected, edit the script, or change its run-time settings.

To add a script:

1 In the Scenario Scripts pane, click the **Add Script** button, or right-click within a column and select **Add Script**. The Add Script dialog box opens.

2 Click **Browse**. The Open Test dialog box opens.

Select the path and file name of the new script. To select a VB Vuser script, browse to locate the **.usr** file.

Note: When you specify the location of a script, you can specify a location that is relative to the current scenario directory. For details, see "Using Relative Paths for Scripts" on page 112.

---

3 Click **Open** to select the files. The Open Test dialog box closes, and the new script name appears in the Add Script dialog box.

4 Click **OK** to close the Add Script dialog box and enter the new script information in the Scenario Scripts pane.

Note: A script's rendezvous points are disabled in a goal-oriented scenario.

Viewing Script Information

Once you have added a script to the list, you can view the details of the
script you selected, edit the script, enable/disable it, or change its run-time
settings.

To view script details:

1 In the Scenario Scripts pane, click the **Details** button, or right-click a script
and select **Details**. The Script Information dialog box opens, displaying the
Name, Path, and Type of the script you selected.
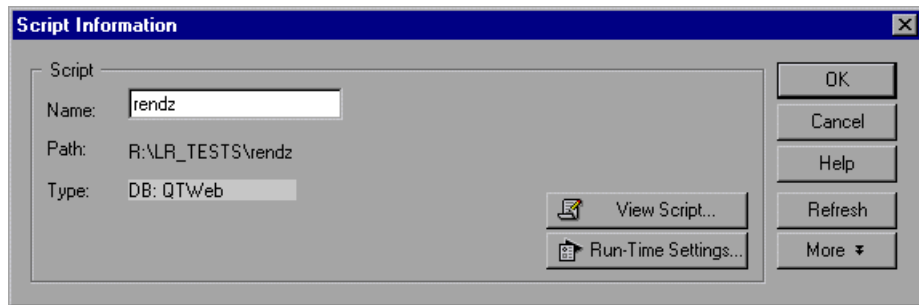
2 Click **Run-Time Settings** to set the script's run-time settings (optional),
which allow you to customize the way the Controller executes a Vuser
script. The Run-Time Settings dialog box opens, displaying the settings you
previously set using VuGen. If you did not set run-time settings for a script
in VuGen, the default VuGen settings are displayed for all but the Log and
Think Time tabs, which display the default Controller settings.

Note: Several protocols, such as Web and Java, have specific settings.

For information on configuring the run-time settings, see the *HP Virtual User Generator User's Guide*.

Note: If you modify the run-time settings from the Controller, LoadRunner runs the script using the modified settings. To restore the initial settings, click **Refresh** and select **Run-Time Settings**.

---

3 To edit the script, click **View Script**. The script generation tool, VuGen, opens. For more information on editing scripts, see the *HP Virtual User Generator User's Guide*.

Note: If you use VuGen to make changes to a script while the Controller is running, click **Refresh** and select **Script** to update the script details in the scenario.

---

4 Click **More** to expand the Script Information dialog box and view additional script information.
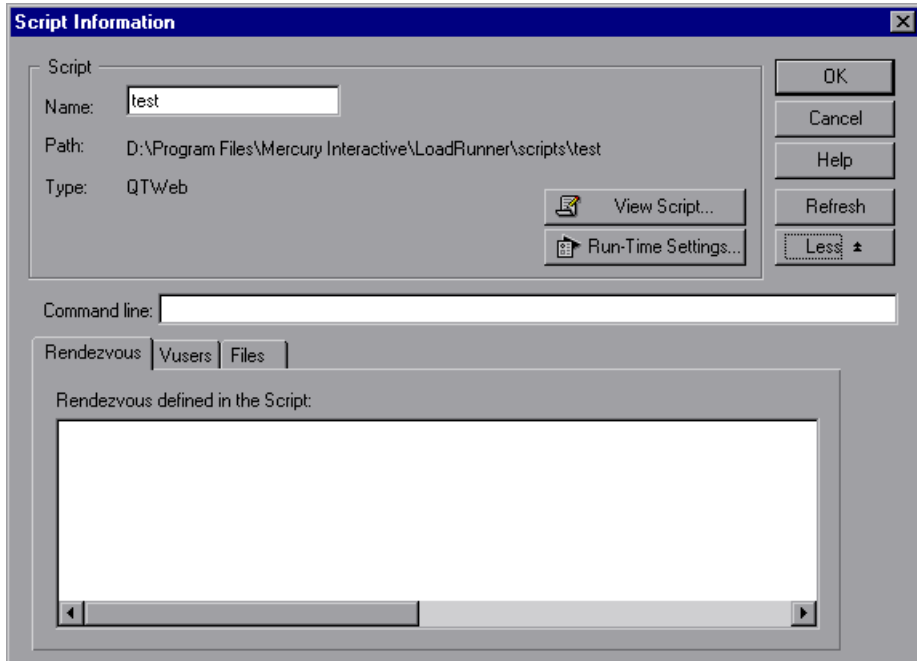
5 In the **Command Line** box, type any command line options to use when running the script. For example: -x value -y value

For information about passing command line argument values to a script, see the *HP Virtual User Generator User's Guide*.

6 To see the rendezvous points included in the selected script, click the **Rendezvous** tab.

7 To see the list of Vusers associated with the selected script, click the **Vusers** tab. If you have not yet created Vusers, the box will be empty.

8 To see the list of files used by the script, select the **Files** tab. By default, this list shows all files in the script's directory (only after your script has been added to the script list). These files include the configuration settings file, the init, run, and end portions of the script, the parameterization definitions file, and the **.usr** file. To add a file to the list, click **Add** and add the file name.

Note: You can delete the files that you add, but not the other files listed.

---

9 Click **OK** to close the Script Information dialog box.

To delete a script:

In the Scenario Scripts pane, click the **Remove Script** button, or right-click the script and select **Remove Script**.

To disable a script:

Click the check box to the left of the Vuser script name. The color of the script entry changes to gray, indicating that the script will not take part in the scenario. To re-enable the Vuser script, click the same box again.

10

Using Rendezvous Points

LoadRunner allows you to check your system's response under specific load. To do this, you can use **rendezvous points** to cause multiple Vusers to perform tasks at exactly the same time, thereby creating intense user load on the server.

This chapter includes:

About Using Rendezvous Points

During a scenario run, you can instruct multiple Vusers to perform tasks simultaneously by using rendezvous points. A rendezvous point creates intense user load on the server and enables LoadRunner to measure server performance under load.

Suppose you want to measure how a Web-based banking system performs when ten Vusers simultaneously check account information. To emulate the required user load on the server, you instruct all the Vusers to check account information at exactly the same time.

You ensure that multiple Vusers act simultaneously by creating a **rendezvous point**. When a Vuser arrives at a rendezvous point, it is held there by the Controller. The Controller releases the Vusers from the rendezvous either when the required number of Vusers arrives, or when a specified amount of time has passed. For details on the release criteria, see "Setting the Rendezvous Policy" on page 205.

You define rendezvous points in the Vuser script. For information about inserting rendezvous points into Vuser scripts, see the *HP Virtual User Generator User's Guide*.

Using the Controller, you can influence the level of server load by selecting:

➤ which of the rendezvous points will be active during the scenario

➤ how many Vusers will take part in each rendezvous

For example, to test a bank server, you could create a scenario that contains two rendezvous points. The first rendezvous ensures that one thousand Vusers simultaneously deposit cash. The second rendezvous ensures that another thousand Vusers simultaneously withdraw cash. If you want to measure how the server performs when only five hundred Vusers deposit cash, you can deactivate (disable) the "withdraw" rendezvous, and instruct only five hundred Vusers to participate in the "deposit" rendezvous.

The following procedure outlines how to control load peaks on the server:

1 Create the Vuser scripts, inserting the necessary rendezvous points.

2 Create a scenario.

When you add a Vuser group to a scenario, LoadRunner scans the group's associated script for the names of the rendezvous points and adds them to the list in the Rendezvous Information dialog box (**Scenario** > **Rendezvous**). If you create another Vuser group that runs the same script, the Controller adds the new Vusers to the rendezvous and updates the list.

3 Set the level of emulated user load.

You determine the exact level of load by selecting the rendezvous points that will take part in the scenario, and how many Vusers will participate in each rendezvous.

4 Set the attributes for the rendezvous (optional).

For each rendezvous you can set **Policy** attributes. For more information, see "Setting the Rendezvous Policy" on page 205.

5 Run the scenario.

Setting the Rendezvous Attributes

You can set the following rendezvous attributes from the Rendezvous Information dialog box (**Scenario** > **Rendezvous**):

➤ Rendezvous Policy

➤ Enabling and Disabling of Rendezvous Points

➤ Enabling and Disabling of Vusers

In addition, the dialog box displays general information about the rendezvous point: which script is associated with the rendezvous, and release history.
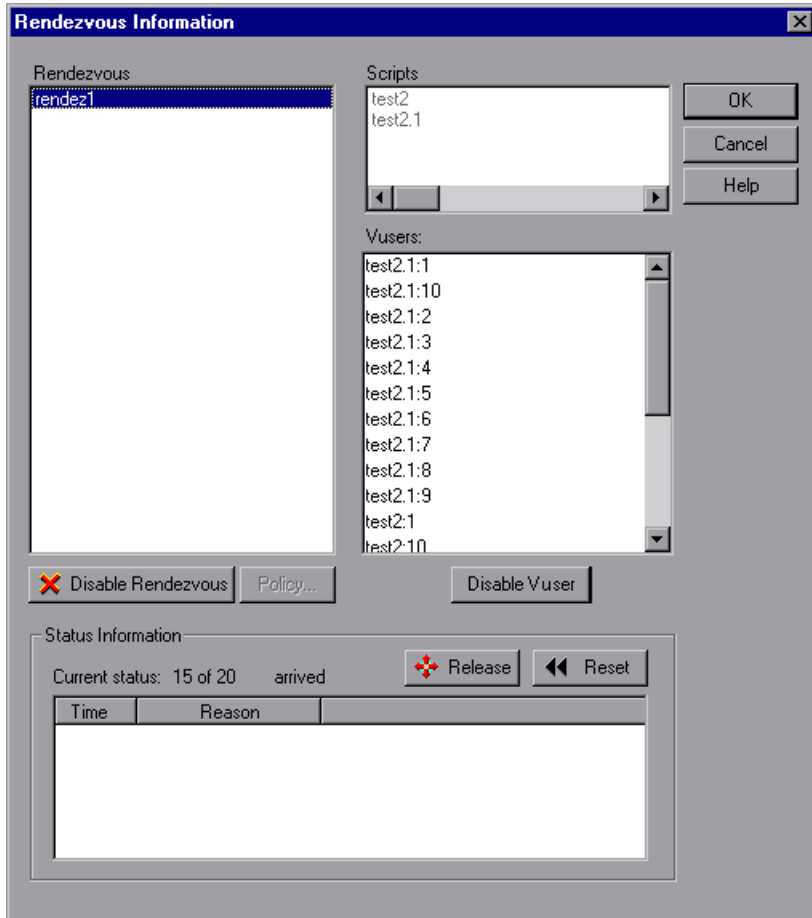
For information on manipulating the Vusers during scenario execution using the Release command, see Chapter 15, "Running a Scenario."

Setting the Rendezvous Policy

Setting the rendezvous policy determines how the Vusers handle a rendezvous point. You set the following policy attributes for each rendezvous:

➤ **Release policy.** How many Vusers will be released from a rendezvous at a time.

➤ **Timeout.** How long the Controller waits before releasing Vusers from a rendezvous.

To set the rendezvous policy attributes:

1 Select **Scenario** > **Rendezvous**. The Rendezvous Information dialog box opens.

2 Select a rendezvous from the **Rendezvous** box, and click the **Policy** button. The Policy dialog box opens.
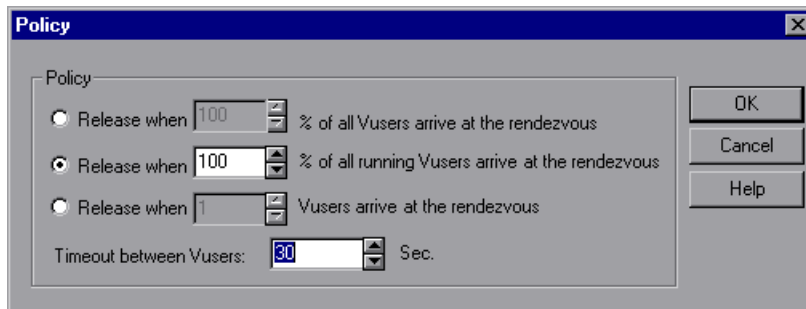
3 In the Policy section, select one of the following three options:

> ➤ **Release when X% of all Vusers arrive at the rendezvous.** Releases the Vusers only when the specified percentage of all Vusers arrives at the rendezvous point.

Note: This option interferes with the scheduling of your scenario. If you select this option, your scenario will not run as scheduled.

> ➤ **Release when X% of all running Vusers arrive at the rendezvous.** Releases the Vusers only when the specified percentage of all Vusers running in the scenario arrives at the rendezvous point.

> ➤ **Release when X Vusers arrive at the rendezvous.** Releases the Vusers only when the specified number arrives at the rendezvous point.

4 Enter a timeout value in the Timeout between Vusers box. After each Vuser arrives at the rendezvous point, LoadRunner waits up to the maximum **timeout** period you set for the next Vuser to arrive. If the next Vuser does not arrive within the **timeout** period, the Controller releases all the Vusers from the rendezvous.

Each time a new Vuser arrives, the timer is reset to zero. The default **timeout** is thirty seconds.

5 Click **OK** to save your settings and close the Policy dialog box.

Enabling and Disabling Rendezvous Points

You can temporarily disable a rendezvous and exclude it from the scenario. By disabling and enabling a rendezvous, you influence the level of server load.

You use the Disable Rendezvous/Enable Rendezvous button in the Rendezvous Information dialog box, to change the status of a rendezvous.

To disable a rendezvous:

1 In the **Rendezvous** box, select the rendezvous you want to disable.

2 Click the **Disable Rendezvous** button. The button changes to **Enable Rendezvous** and the rendezvous becomes disabled.

To enable a rendezvous:

1 In the **Rendezvous** box, select the disabled rendezvous that you want to enable.

2 Click the **Enable Rendezvous** button. The button changes to **Disable Rendezvous** and the rendezvous becomes enabled.

Enabling and Disabling Vusers at Rendezvous Points

In addition to disabling a rendezvous point for all Vusers in a scenario, LoadRunner lets you disable it for specific Vusers. By disabling Vusers at a rendezvous, you temporarily exclude them from participating in the rendezvous. Enabling disabled Vusers returns them to the rendezvous. You use the Disable and Enable commands to specify which Vusers will take part in a rendezvous.

To disable a Vuser in a rendezvous:

1 In the **Rendezvous** box, select the rendezvous for which you want to disable Vusers.

2 In the **Vusers** box, select the Vusers you want to exclude from the rendezvous. Select multiple Vusers using the CTRL key.
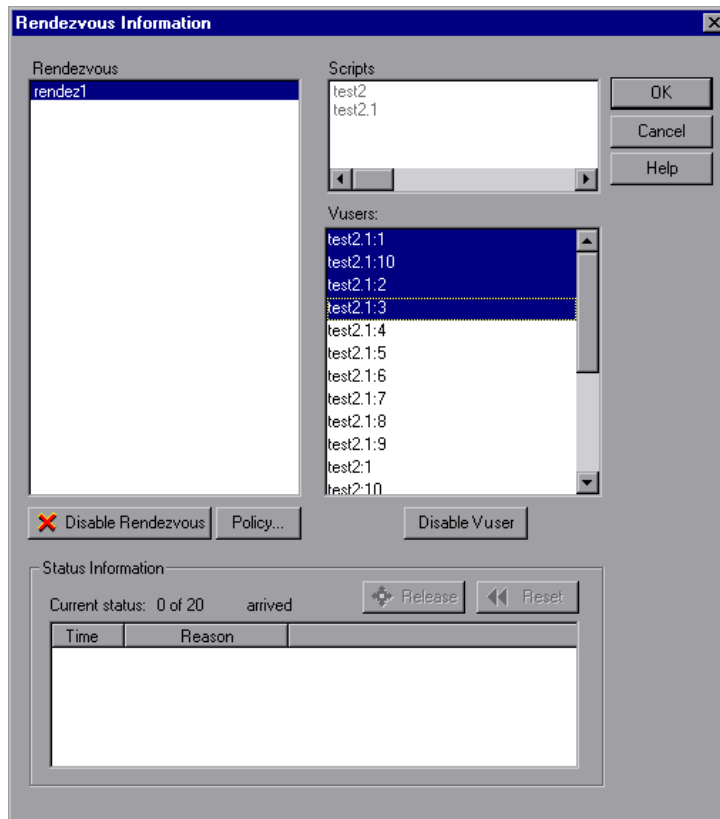
3 Click the **Disable Vuser** button below the Vusers box. The disabled Vusers change from black to gray and will not take part in the rendezvous.

To enable a Vuser, select it and click **Enable Vuser**.

Viewing Rendezvous Information

During and after a scenario, you can view the rendezvous status in the Rendezvous Information dialog box. The following information is provided:

➤ **Current Status.** The number of Vusers that arrived at the rendezvous point out of the total number of Vusers assigned to the rendezvous.

➤ **Time.** The time at which the Vusers at the rendezvous point were released.

➤ **Reason.** The reason the Vusers at the rendezvous point were released. The possible reasons are **Timeout** or **Arrived**.

To view rendezvous information:

Select the rendezvous whose information you want to view. The rendezvous status is displayed in the Status Information section.
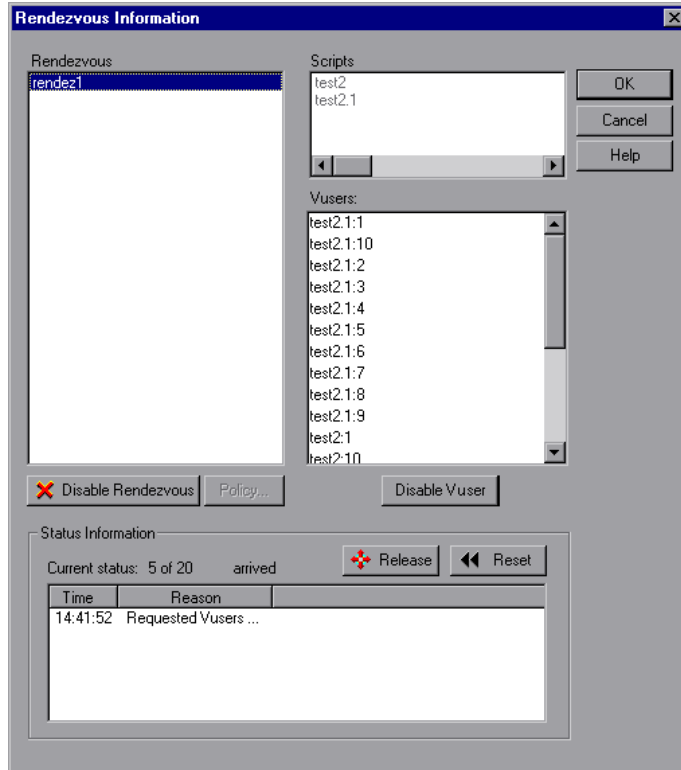
Understanding the Rendezvous Information Dialog Box

The Rendezvous Information dialog box enables you to view and modify the attributes of each rendezvous point in the scenario.

➤ **Rendezvous.** Displays the names of the rendezvous points in the scenario.

➤ **Enable Rendezvous/Disable Rendezvous.** Enables or disables the selected rendezvous points from participating in the scenario.

➤ **Scripts.** Lists the Vuser scripts that are associated with the rendezvous points.

➤ **Vusers.** Lists the Vusers that are associated with the rendezvous points.

    ➤ **Enable Vuser/Disable Vuser.** Enables or disables a Vuser from taking part in the rendezvous.

➤ **Policy.** Opens the Policy dialog box, enabling you to set how many Vusers are released from a rendezvous at a time, as well as the amount of time the Controller waits before releasing Vusers from a rendezvous.

    ➤ **Timeout.** Enter the timeout value (in seconds). After each Vuser arrives at the rendezvous point, LoadRunner waits up to the number of timeout seconds specified for the next Vuser to arrive. If the next Vuser does not arrive within the timeout period, the Controller releases all the Vusers from the rendezvous. Each time a new Vuser arrives, the timer is reset to zero. The default timeout is thirty seconds. You set a timeout for each rendezvous point.

➤ **Status Information**

    ➤ **Current Status.** Displays the number of Vusers that arrived at the rendezvous point out of the total number of Vusers assigned to the rendezvous.

    ➤ **Time.** Displays the time at which the rendezvous was released.

    ➤ **Reason.** Displays the reason for the release of the Vusers from the rendezvous point. The possible reasons are **Timeout** or **Arrived**.

    ➤ **Release.** Releases all Vusers currently waiting at the selected rendezvous point. If you want the scenario to continue even though all the Vusers did not reach the rendezvous, click this button.

    ➤ **Reset.** Resets the Status Information, removing the information currently displayed.

11

Configuring a Scenario

You can configure how load generators and Vusers behave when you run a scenario so that the scenario accurately emulates your working environment.

This chapter includes:

About Configuring a Scenario

Before you run a scenario, you can configure both the load generator and Vuser behaviors for the scenario. Although the default settings correspond to most environments, LoadRunner allows you to modify the settings to customize the scenario behavior. The settings apply to all future scenario runs and generally need to be set only once.

The settings described in this chapter apply to all the load generators in a scenario. To change the settings for individual load generators, see Chapter 5, "Creating Manual Scenarios." If the global scenario settings differ from those of an individual load generator, the load generator settings override them.

The settings discussed in this chapter are unrelated to the Vuser run-time settings. These settings, which apply to individual Vusers or scripts, contain information about logging, think time, and the network, the number of iterations, and the browser. For information on setting the run-time settings, see the *HP Virtual User Generator User's Guide*.

For information on setting the options for online monitors, see Chapter 24, "Online Monitoring."

The LoadRunner Expert mode allows you to configure additional settings for the LoadRunner agent and other LoadRunner components. For more information, see Appendix C, "Working in Expert Mode."

Setting Timeout Intervals

The Timeout tab lets you specify timeout values for certain commands related to the load generator. If the command is not executed successfully within the timeout period, the load generator status changes to ERROR.

To set timeout intervals:

 1 Select **Tools** > **Options**. The Options dialog box opens. Select the **Timeout** tab.
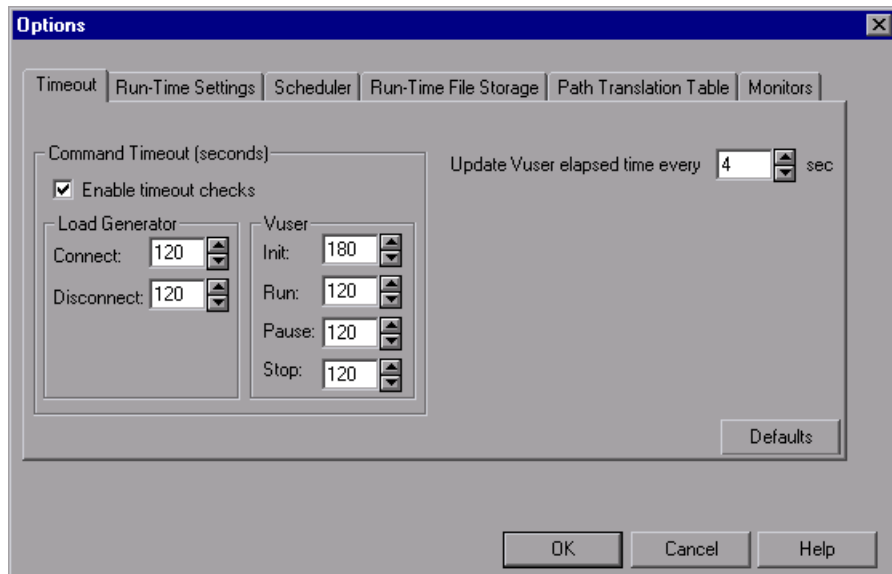
 2 To specify a command timeout interval, select the **Enable timeout checks** check box and specify the appropriate timeouts. Clear the **Enable timeout checks** check box to disable the timeout test.

 3 Specify the frequency at which LoadRunner updates the elapsed time, in the **Update Vuser elapsed time every** box.

Understanding the Options - Timeout Tab

LoadRunner enables you to set the timeout interval for commands and Vuser elapsed time.

The command timeouts are the maximum time limits for various LoadRunner commands. When a command is issued by the Controller, you set a maximum time for the load generator or Vuser to execute the command. If it does not complete the command within the timeout interval, the Controller issues an error message.

➤ **Command Timeout (seconds)**

  ➤ **Enable timeout checks.** Instructs LoadRunner to monitor the status of load generators and Vusers after a command is issued by the Controller. If the load generator or Vuser does not complete the command within the timeout interval you specified, the Controller issues an error message. If you disable the timeout limitations, LoadRunner waits an unlimited time for the load generators to connect and disconnect, and for the Initialize, Run, Pause, and Stop commands to be executed.

  ➤ **Load Generator**

    ➤ **Connect.** Enter the time limit that LoadRunner waits to connect to any load generator. If a connection is not successful within this time, the status of the load generator changes to FAILED. The default connection timeout is 120 seconds.

    ➤ **Disconnect.** Enter the time limit that LoadRunner waits to disconnect from any load generator. If a disconnection is not successful within this time, the status of the load generator changes to FAILED. The default disconnection timeout is 120 seconds.

Note: LoadRunner recognizes the fact that the number of active Vusers influences the timeout values. For example, 1000 Vusers trying to initialize will take much longer than 10 Vusers. LoadRunner adds an internal value, based on the number of active Vusers, to the specified timeout value.

➤ **Vuser**

> ➤ **Init.** Enter the timeout value for the Initialize command. The default time limit is 180 seconds.

> ➤ **Run.** Enter the timeout value for the Run command. The default time limit is 120 seconds.

> ➤ **Pause.** Enter the timeout value for the Pause command. The default time limit is 120 seconds.

> ➤ **Stop.** Enter the timeout value for the Stop command. The default time limit is 120 seconds.

➤ **Update Vuser elapsed time every.** Specifies the frequency at which LoadRunner updates the value displayed in the Elapsed Time column in the Vusers dialog box. The default is 4 seconds.

For example:

If you select a Vuser and click the **Initialize** button, LoadRunner checks whether the Vuser reaches the **READY** state within 180 seconds (the default **Init** timeout period); if it does not, the Controller issues a message indicating that the **Init** command timed out.

Configuring Scenario Run-Time Settings

You can specify scenario run-time settings relating to Vuser quotas, stopping Vusers, and random sequence seed using the Run-Time Settings tab.

To set the scenario run-time settings:

 1 Select **Tools** > **Options**. The Options dialog box opens. Click the **Run-Time Settings** tab.
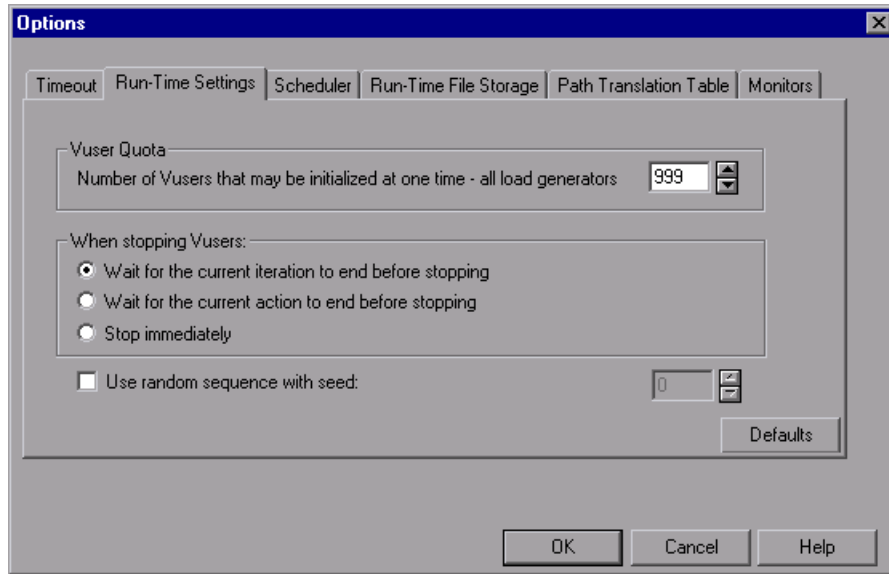
 2 To set a Vuser quota, specify the desired value.

 3 Select the way in which you want LoadRunner to stop running Vusers.

 4 To specify a seed value for a random sequence, select the **Use random sequence with seed** check box and enter the desired seed value.

Understanding the Options - Run-Time Setting Tab

The Run-Time Setting Tab lets you specify run-time setting values such as the Vuser quota, the way in which Vusers are stopped, or the seed for random sequences.

➤ **Vuser Quota**. To prevent your system from overloading, you can set quotas for Vuser activity. The Vuser quotas apply to Vusers on all load generators.

➤ **Number of Vusers that may be initialized at one time - all load generators.** Sets the maximum number of Vusers that the load generator can initialize at a time (when you send an Initialize command).

➤ **When stopping Vusers.** Lets you control the way in which Vusers stop running when you click the Stop button.

Select one of the following options:

➤ **Wait for the current iteration to end before stopping.** Instructs LoadRunner to allow a Vuser to complete the iteration it is running before stopping. The Vusers move to the GRADUAL EXITING status and exit the scenario gradually.

➤ **Wait for the current action to end before stopping.** Instructs LoadRunner to allow a Vuser to complete the action it is running before stopping. The Vusers move to the GRADUAL EXITING status and exit the scenario gradually.

➤ **Stop immediately.** Instructs LoadRunner to stop running the Vusers immediately. The Vusers move to the EXITING status and exit the scenario immediately.

➤ **Use random sequence with seed.** Allows LoadRunner to use a seed number for random sequencing. Each seed value represents one sequence of random values used for test execution. Whenever you use this seed value, the same sequence of values is assigned to the Vusers in the scenario. This setting applies to parameterized Vuser scripts using the Random method for assigning values from a data file. It will also affect the random percentage of recorded think time (see information on the Run-Time Settings dialog box in the *HP Virtual User Generator User's Guide*).

Enable this option if you discover a problem in the test execution and want to repeat the test using the same sequence of random values.

Setting the Run-Time File Location

When you run a scenario, by default the run-time files are stored locally on each Vuser load generator (the machine running the Vuser script). The default location of the files is under the temporary directory specified by the load generator's environment variables (on Windows, TEMP or TMP, and on UNIX, $TMPDIR or $TMP). If no environment variable is defined, the files are saved to the /tmp directory.

Note: The run-time file storage settings that are described in this chapter apply to all the load generators in a scenario. You can change the settings for individual load generators as described in "Configuring Load Generators" on page 84.

The primary run-time files are as follows:

Run-Time File Type	Description
Vuser Script files	When you run a Vuser, the Controller sends a copy of the associated Vuser script to the Vuser load generator. The script is stored in the load generator's temporary run-time directory.
Result files	While you run a scenario, the participating Vusers write their results to the temporary run-time file directory. After scenario execution, these result files are collated or consolidated—results from all of the load generators are transferred to the results directory. You set the location of the results directory as described in Chapter 15, "Running a Scenario." After collating the results, the temporary run-time directory is deleted.

To specify where LoadRunner stores run-time files:

1 Select **Tools** > **Options**. The Options dialog box opens. Select the **Run-Time File Storage** tab.
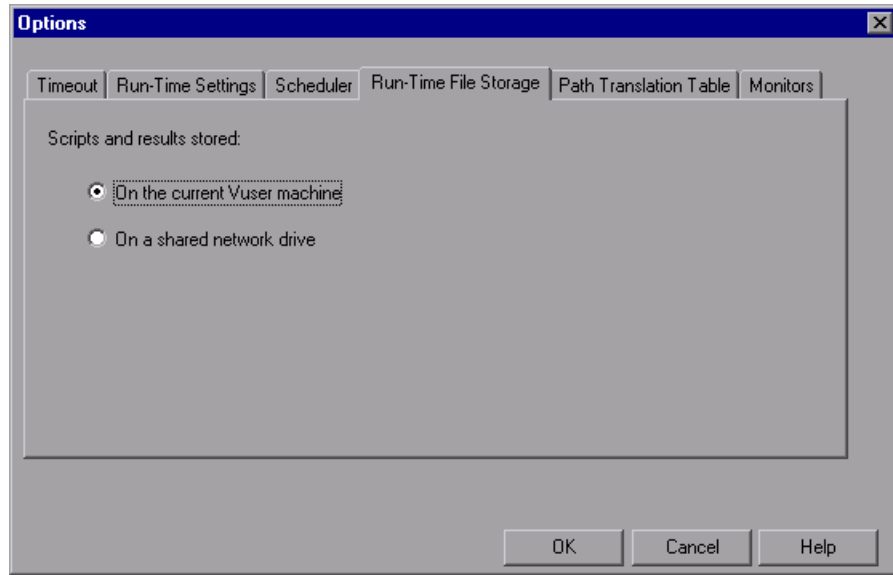
By default, the **On the current Vuser machine** option is selected. This means that all run-time files—including result files and script files—are stored on the Vuser load generators. The only exception is for Vusers running on the local load generator (Controller machine), where you must use the shared drive option.

2 To store script and result files on a shared network drive, click **On a shared network drive**. To set the exact location on the network drive, see Chapter 12, "Preparing to Run a Scenario."

3 Click **OK** to close the dialog box.

Understanding the Options - Run-Time File Storage Tab

The Run-Time File Storage tab lets you specify where LoadRunner saves runtime files.

➤ **Scripts and results stored.** Select one of the following options:

➤ **On the current Vuser machine.** Instructs the Controller to save the run-time files on the computer that is running the Vuser script. On an NT-based computer, the results are saved to the directory defined by the TEMP or TMP environment variables. On a UNIX machine, the results are saved to the directory defined by the TMPDIR environment variable. If the TMPDIR environment variable is not defined, the results are saved to the **/tmp** directory.

Note: If you select to save result files on the Vuser load generators, you must collate the results before you can perform any analysis. You can wait for LoadRunner to collate the results when you launch HP LoadRunner Analysis, or you can collate results by choosing **Results** > **Collate Results**. Alternatively, select **Results** > **Auto Collate Results** to automatically collate the results at the end of each scenario run.

➤ **On a shared network drive.** Instructs the Controller to save the scenario results and/or the Vuser scripts on a shared network drive. A shared network drive is a drive to which the Controller and all the load generators in the scenario have read and write permission. If you select to save results to a shared network drive, you may need to perform path translation. Path translation ensures that the specified results directory is recognized by the remote load generator. For information about path translation see Appendix B, "Performing Path Translation."

If you specify that all Vusers access their Vuser scripts directly at some shared location, no transfer of script files occurs at run time. This alternative method may be useful in either of the following situations:

➤ The file transfer facility does not work.

➤ The Vuser script files are large and therefore take a long time to transfer. Remember that Vuser script files are transferred only once during a scenario.

This alternate method often necessitates path translation. For details, see Appendix B, "Performing Path Translation."

Specifying Path Translation

If you specified a shared network drive for run-time file storage (see "Setting the Run-Time File Location" on page 220), you may need to perform **path translation**. Path translation is a mechanism used by LoadRunner to convert remote path names. A typical scenario may contain several load generators that map the shared network drive differently. For more information, see Appendix B, "Performing Path Translation."

12

Preparing to Run a Scenario

Before you run a scenario, you specify a location for the scenario results and other run-time related settings.

This chapter includes

About Preparing to Run a Scenario

Before you run a scenario, you need to specify the location of the results (mandatory), assign a name to the results, schedule the scenario, and provide scenario summary information. In addition, you can specify the applications to invoke at the start of a scenario.

Although most of the pre-scenario settings are optional, using them can enhance the testing process. These values are scenario-specific—you can set different values for each LoadRunner scenario.

For information on one-time configuration settings such as timeout, output, and quotas, see Chapter 11, "Configuring a Scenario."

Specifying a Results Location

When you run a scenario, by default the run-time files are stored locally on each load generator. After the scenario, the results are collated together and processed on the Controller machine. Alternatively, you can instruct LoadRunner to save the results on a shared network drive. For information about specifying a file storage method, see the Run-Time File Storage settings in Chapter 11, "Configuring a Scenario."

LoadRunner allows you to give descriptive names to each result set. This is especially useful for cross results analysis, in which LoadRunner superimposes the results of several scenario runs in a single graph and lets you compare the results of multiple scenario runs. The descriptive graph names enable you to distinguish between the results of the multiple runs.

In the example below, the results of two scenario runs are superimposed. The result sets are **res12**, and **res15**.
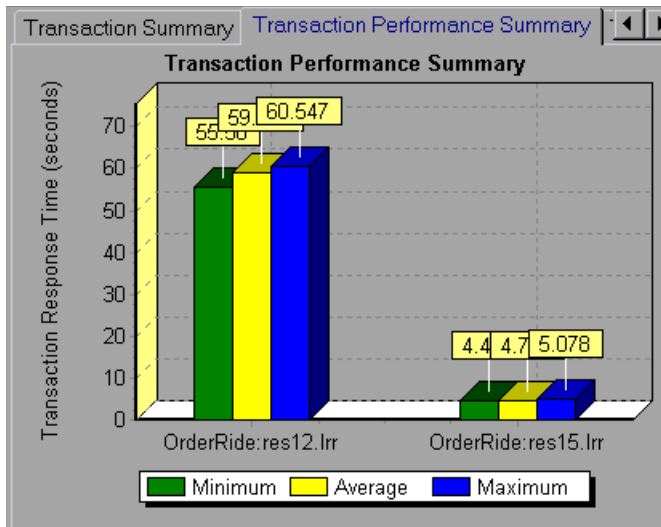
For more details on cross result graphs, see the *HP LoadRunner Analysis User's Guide*.

Note: You can also use HP's Web-based test management program, Quality Center, to store results to a project. For information, see Chapter 13, "Managing Scenarios Using Quality Center."

To specify where results are stored:

1 Select **Results** > **Results Settings**. The Set Results Directory dialog box opens.
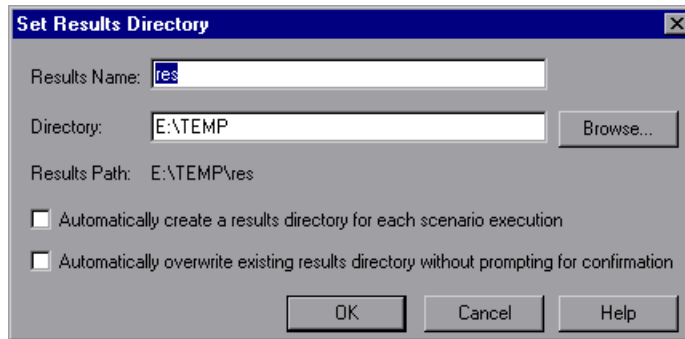
2 In the Results Name box, enter a name for the results. Avoid using the same name with different paths, since the names will appear identical on the graphs.

3 In the Directory box, type the full path of the results directory. If you are using the default file storage setting (local machine), specify a directory in which to store all of the collated results after the scenario run. If you specified a shared network drive as the file storage method, specify the directory to which Vuser groups should write during scenario execution.

4 Select the appropriate check box for subsequent executions:

   ➤ **Automatically create a results directory for each scenario execution**

   ➤ **Automatically overwrite existing results directory without prompting for confirmation**.

5 Click **OK** to save the results directory setting.

Understanding the Set Results Directory Dialog Box

The Set Results Directory dialog box lets you set the location in which the Controller saves scenario results.

Note: If you have an open connection to a Quality Center project, the Controller saves the results to a test set. You can also save the results directly to disk using the standard file system.

➤ **Results Name.** Specify a name for the results. The Controller saves the results using the name that you specify.

➤ **Directory.** Specify a location in the file system where the Controller will save the results. Click **Browse** to find the desired location. The Controller creates a subdirectory in the results directory. All results are saved within this subdirectory.

➤ **Results Path.** Displays the specified location for the results.

➤ **Automatically create a results directory for each scenario execution.** Instructs LoadRunner to create a unique results directory for each scenario execution. By default, the result names are **res1**, **res2**, **res3**, etc.

➤ **Automatically overwrite existing results directory without prompting for confirmation.** Instructs LoadRunner to automatically overwrite previous result sets, without prompting the user.

➤ **Quality Center** (only when connected to Quality Center). Enables you to save the results to a Quality Center test set.

➤ **File System** (only when connected to Quality Center). Displays the default LoadRunner directory path.

Results Directory File Structure

When you set the results directory, you also specify a results name. LoadRunner creates a subdirectory using the results name, and places all of the data it gathers in that directory. Every set of results contains general information about the scenario in a result file (**.lrr**) and an event (**.eve**) file.

During scenario execution, LoadRunner creates a directory for each group in the scenario and a subdirectory for each Vuser. A typical result directory has the following structure:
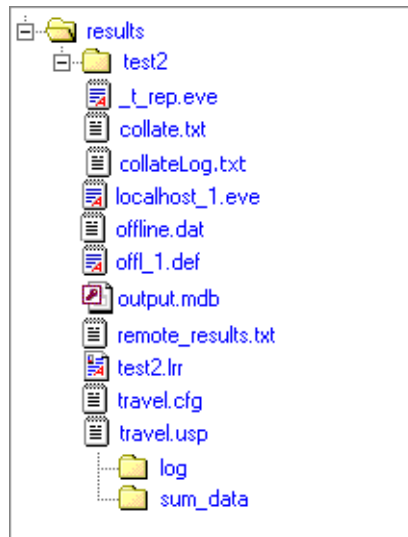
The following directories/files are represented in the tree:

Directory/File	Description
Results	Results directory
Test2	Results name
t_rep.eve	In the main result directory; contains Vuser and rendezvous information
collate.txt	Contains the file paths of the result files, and Analysis collation information

Directory/File	Description
collateLog.txt	Contains the status (succeeded, failed) of result, diagnostics, and log file collation from each load generator
local_host.eve	Contains information from each agent host
offline.dat	Contains sample monitor information
***.def files**	Definition files for graphs that describe the online and other custom monitors
output.mdb	The database created by Analysis (from the results files) that stores the output information
remote_results.txt	Contains the file paths for the host event files
results_name.lrr	The Analysis document file
***.cfg files**	Contain a listing of the script's run-time settings as defined in the VuGen application (think time, iterations, log, Web)
***.usp files**	Contain the script's run logic, including how the actions sections run
log directory	➤ Contains output information generated during replay for each Vuser ➤ A separate directory exists for each Vuser group that runs in the scenario ➤ Each group directory consists of Vusers subdirectories
sum_data directory	A directory containing the graph summary data (**.dat**) files

When you generate analysis graphs and reports, the Analysis engine copies all of the scenario result files (**.eve** and **.lrr**) to a database. Once the database is created, Analysis works directly with the database and does not use the result files.

For information on Analysis, see the *HP LoadRunner Analysis User's Guide*.

Collating Results

When you run a scenario, by default all Vuser information is stored locally on each load generator. After scenario execution, the results are automatically collated or consolidated—results from all of the load generators are transferred to the results directory. You set the location of the results directory as described in "Specifying a Results Location" on page 226.

Note: If you have selected to store all the scenario results directly to a shared network drive, then collation of the results is not required. See "About Configuring a Scenario" on page 213 for details on changing how results are stored.

To enable/disable automatic collation:

Select **Results** > **Auto Collate Results** to enable automatic collation. When this feature is enabled, the Auto Collate Results icon is displayed in the status bar. To disable automatic collation, clear the check mark adjacent to the option.

Note: If you are using the diagnostics modules, results must be collated even if the load generator is localhost.

To manually collate results:

Select **Results** > **Collate Results** > **Collate Results**. The Collate Results dialog box opens, and displays the progress of result, diagnostics, and log file collation from each load generator.
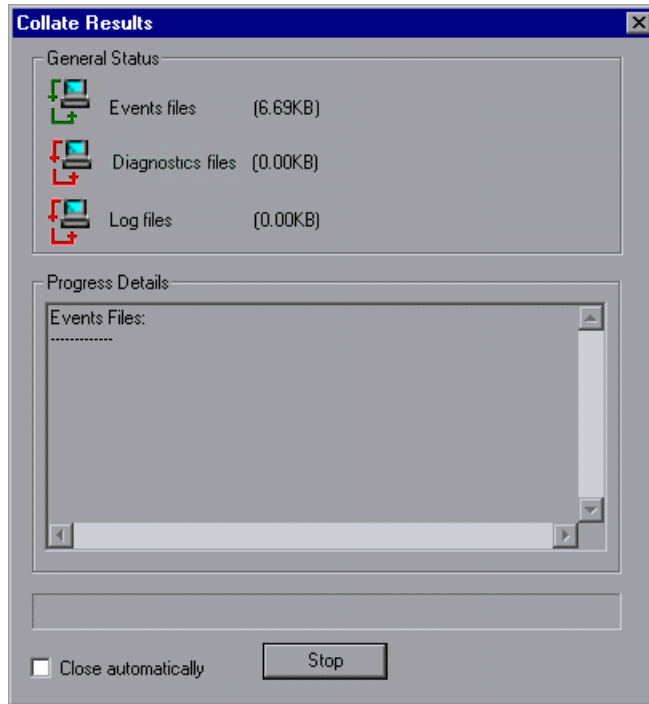
➤ **General Status.** The Collate Results files display the status and file size of the Event, Diagnostics, and Log files.

Note: The file size shown is before compression. After compression, the result files are smaller in size.

➤ **Progress Details.** Displays the status (succeeded, failed) of result, diagnostics, and log file collation from each load generator. This information is stored in the **collateLog.txt** file.

➤ **Close automatically.** Automatically close the Collate Results dialog box after collation is completed.

To stop collating the results:

Click **Stop** and then **Close.** To resume collating the results, select **Results** > **Collate Results** > **Continue stopped collation**.

Note: You can choose to disable log file collation. For more information, see "Options - General Settings" on page 452.

The log and result directories are only deleted from a load generator once LoadRunner successfully collates the results from the machine. You can therefore close the Controller after saving a scenario, and collate the results once you reopen the scenario in the Controller.

If collation fails due to a lack of disk space, select **Results** > **Collate Results** > **Recollate**. LoadRunner attempts to collate the results again, without compressing the **.eve** file.

Before generating the analysis data, LoadRunner automatically collates the results if they were not previously collated.

Note: If you enabled the **Auto Load Analysis** option in the Results menu, Analysis may open during a lengthy collation process, displaying Analysis summary data.

233

Setting Scenario Summary Information

The Controller allows you to provide a detailed description of the scenario. In addition, you can specify the author's name and a subject title for the scenario. Whenever you open this scenario, the summary information is available to you.

You can open the Summary Information dialog box from **Scenario** > **Summary Information**.
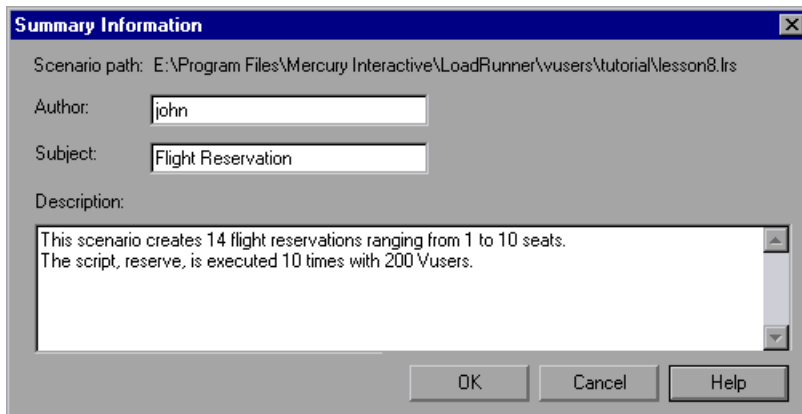
➤ **Scenario Path.** Displays the name and location of the scenario definition file (**.lrs**).

➤ **Author.** Enter the name of the scenario's author.

➤ **Subject.** Enter a subject name or short title for the scenario.

➤ **Description.** Enter a description of the scenario.

13

Managing Scenarios Using Quality Center

LoadRunner's integration with Quality Center lets you manage LoadRunner scenarios using Quality Center. Quality Center helps you organize and manage your scripts, scenarios, and results.

This chapter includes:

➤ About Managing Scenarios Using Quality Center on page 235

➤ Connecting to and Disconnecting from Quality Center on page 236

➤ Opening Scenarios from a Quality Center Project on page 240

➤ Saving Scenarios to a Quality Center Project on page 241

➤ Saving Results to a Quality Center Project on page 243

➤ Adding Vuser Scripts from a Quality Center Project on page 244

About Managing Scenarios Using Quality Center

The Controller works together with Quality Center, HP's Web-based test management tool. HP Quality Center provides an efficient method for storing and retrieving Vuser scripts, scenarios, and results. You store scenarios and results in a Quality Center project and organize them into unique groups.

In order for the Controller to access a Quality Center project, you must connect it to the Web server on which Quality Center is installed. You can connect to either a local or remote Web server.

For more information on working with Quality Center, refer to the *Quality Center User's Guide*.

Connecting to and Disconnecting from Quality Center

To store and retrieve scenarios and results from Quality Center, you need to connect to a Quality Center project. You can connect or disconnect from a Quality Center project at any time during the testing process.

Connecting LoadRunner to Quality Center

The connection process has two stages. First, you connect to a local or remote Quality Center Web server. This server handles the connections between the Controller and the Quality Center project.

Next, you choose the project you want to access. The project stores the scenarios and results that you created in the Controller.

Note: Quality Center projects are password protected, so you must provide a user name and a password.

To connect to Quality Center:

 1 Select **Tools** > **Quality Center Connection**.

 2 The Quality Center Connection - Server Connection dialog box opens.
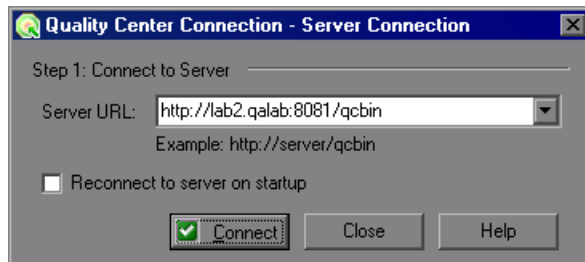
3 In the **Server URL** box, type the URL address of the Web server where Quality Center is installed.

Note: You can choose a Web server accessible via a Local Area Network (LAN) or a Wide Area Network (WAN).

---

4 To automatically reconnect to the Quality Center server the next time you open the Controller, select the **Reconnect to server on startup** check box.

5 Click **Connect**. The Quality Center Connection dialog box opens.

After the connection to the server is established, the server's name is displayed in read-only format in the Server box.

6 Authenticate your user information:

   a In the **User name** box, type your Quality Center user name.

   b In the **Password** box, type your Quality Center password.

c Click **Authenticate** to authenticate your user information against the Quality Center server.

After your user information has been authenticated, the fields in the Authenticate user information area are displayed in read-only format. The **Authenticate** button changes to a **Change User** button.

You can log in to the same Quality Center server using a different user name by clicking **Change User**, entering a new user name and password, and then clicking **Authenticate** again.

7 If you selected **Reconnect to server on startup** above, the **Authenticate on startup** option is enabled. To authenticate your user information automatically, the next time you open the Controller, select **Authenticate on startup**.

8 Enter the details for logging in to the project:

a In the **Domain** box, select the domain that contains the Quality Center project. Only those domains containing projects to which you have permission to connect to are displayed. (If you are working with a project in versions of TestDirector earlier than version 7.5, the **Domain** box is not relevant. Proceed to the next step.)

b In the **Project** box, enter the Quality Center project name or select a project from the list. Only those projects that you have permission to connect to are displayed.

c Click **Login**.

9 To log in to the selected project on startup, select **Login to project on startup**. This option is only enabled when the **Authenticate on startup** check box is selected.

10 Click **Close** to close the Quality Center Connection dialog box.

Disconnecting LoadRunner from Quality Center

You can disconnect The Controller from a selected Quality Center project and Web server.

To disconnect from Quality Center:

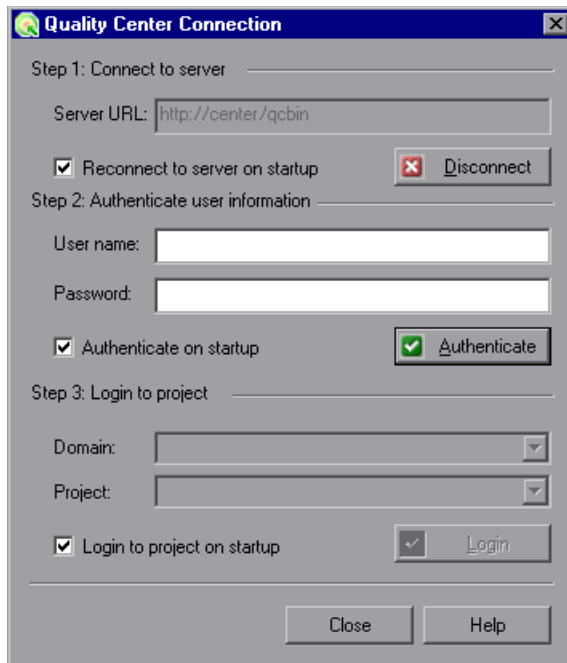**1** Select **Tools** > **Quality Center Connection**. The Quality Center Connection dialog box opens.
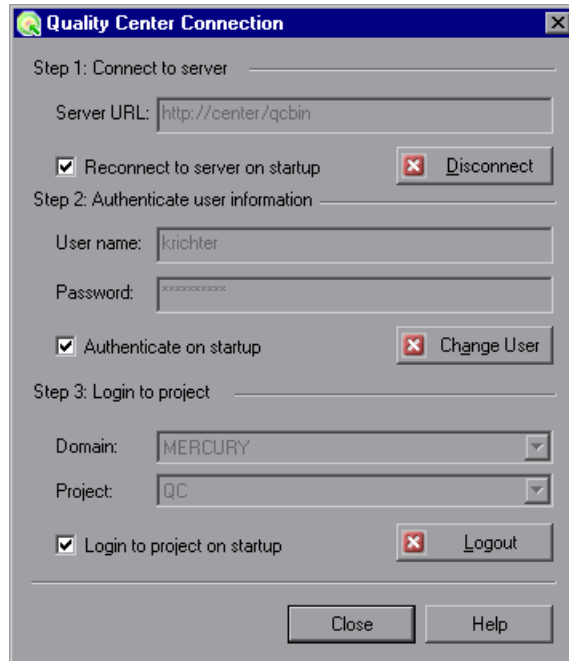
2 To disconnect from the selected project, under **Login to project**, click **Logout**. If you want to open a different project while using the same server, enter the Quality Center project name in the **Project** box or select a project from the list.

3 To disconnect from the Quality Center server, under **Connect to server**, click **Disconnect**.

4 Click **Close** to close the Quality Center Connection dialog box.

Opening Scenarios from a Quality Center Project

When connected to a Quality Center project, you can open your scenarios from Quality Center. You locate tests according to their position in the test plan tree, rather than a location in the file system.

To open a scenario from a Quality Center project:

1 Connect to the Quality Center server. (**Tools** > **Quality Center Connection**)

2 Choose **File** > **Open**. The Open Scenario from Quality Center Project dialog box opens and displays the test plan tree.
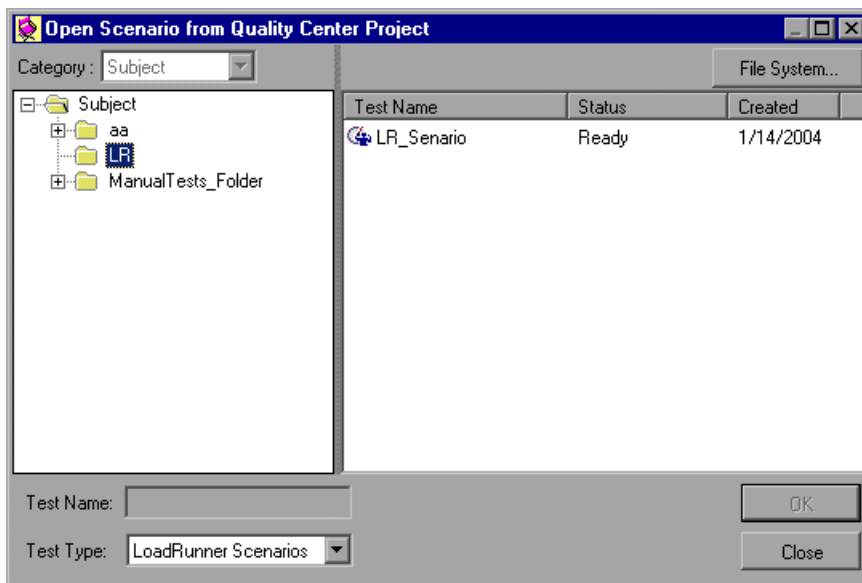
Note: To open a scenario directly from the file system, click the **File System** button. (To return to the Open from Quality Center Project dialog box, click the **Quality Center** button.)

3 Click the relevant subject in the test plan tree. To expand the tree and view sublevels, double-click closed folders. To collapse the tree, double-click open folders.

Note that when you select a subject, the scenarios that belong to the subject appear in the Test Name list.

4 Select a scenario from the Test Name list. The scenario appears in the read-only Test Name box.

5 Click **OK** to open the scenario. The Controller loads the scenario and displays its name in the title bar.

Saving Scenarios to a Quality Center Project

When The Controller is connected to a Quality Center project, you can create new scenarios in The Controller and save them directly to your project. To save a scenario, you give it a descriptive name and associate it with the relevant subject in the test plan tree. This helps you to keep track of the scenarios created for each subject and lets you view the progress of test planning and creation.

To save a scenario to a Quality Center project:

1 Connect to the Quality Center server. (**Tools** > **Quality Center Connection**)

2 Choose **File** > **Save As**. The Save Scenario to Quality Center Project dialog box opens and displays the test plan tree.
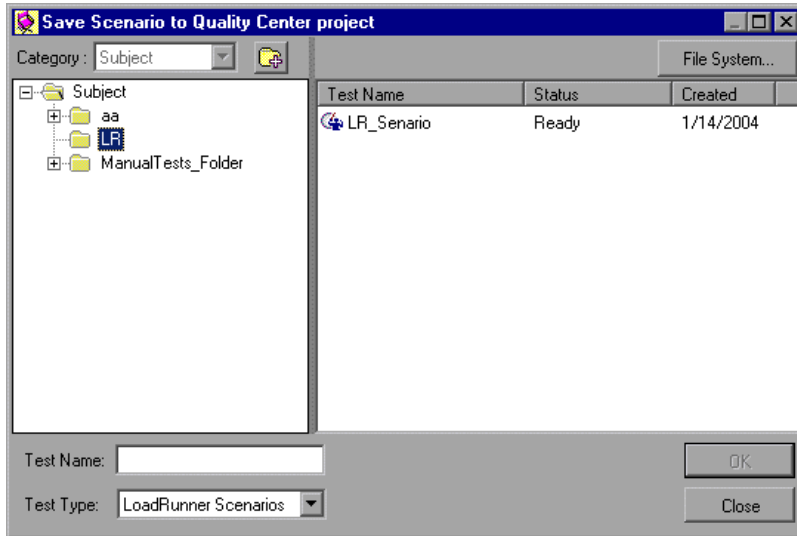
To save a scenario directly in the file system, click the **File System** button. The Save Scenario dialog box opens. (From the Save Scenario dialog box, you may return to the Save Scenario to Quality Center Project dialog box by clicking the **Quality Center** button.)

3 Select the relevant subject in the test plan tree. To expand the tree and view a sublevel, double-click a closed folder. To collapse a sublevel, double-click an open folder.

4 In the Test Name box, enter a name for the scenario. Use a descriptive name that will allow you identify the scenario easily.

5 Click **OK** to save the scenario and close the dialog box.

The next time you start Quality Center, the new scenario will appear in Quality Center's test plan tree.

Saving Results to a Quality Center Project

Before you run a scenario, you set the results location. When LoadRunner is connected to a Quality Center project, results are saved to a test set. You can also save the results to disk using the standard file system.

To save results to a Quality Center project:

 1 Connect to the Quality Center server (see "Connecting LoadRunner to Quality Center" on page 236).

 2 In the Controller, select **Results** > **Results Settings**. The Set Results Directory dialog box opens.
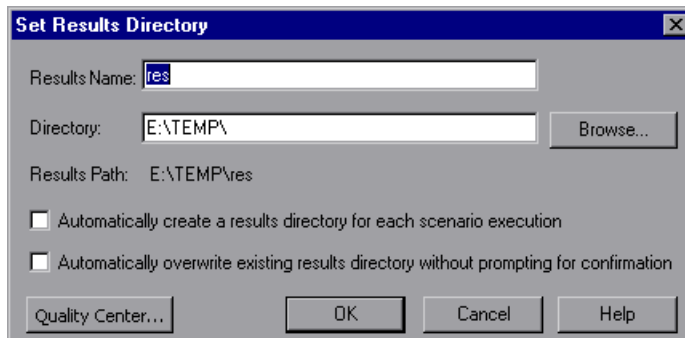
 3 Click **Quality Center**. The **Directory** box changes to **Test Sets**.
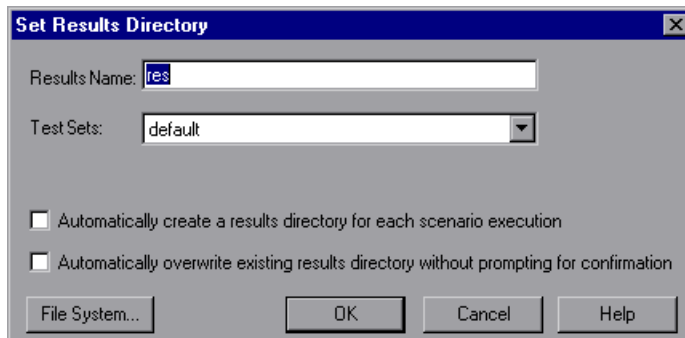
 4 In the Results Name box, enter a name for the results.

 5 In the Test Sets list, accept the default test set name or select a different name.

6 Select the appropriate check box:

➤ **Automatically create a results directory for each scenario execution.**
   Instructs LoadRunner to create a unique results directory for each
   scenario execution. By default, the result names are res1, res2, res3, etc.

➤ **Automatically overwrite existing results directory without prompting for
   confirmation.** Instructs LoadRunner to automatically overwrite previous
   result sets, without prompting the user.

7 Click **OK** to save the results settings.

Adding Vuser Scripts from a Quality Center Project

You can add Vuser scripts from a Quality Center project to the Controller's
script list. You can add the script to either a manual or a goal-oriented
scenario.

This section includes:

➤ "Adding a Vuser Script to a Manual Scenario" on page 245

➤ "Adding a Vuser Script to a Goal-Oriented Scenario" on page 246

Adding a Vuser Script to a Manual Scenario

When you create a manual scenario, you use the Add Group dialog box to add Vuser scripts.

To add a Vuser script to a manual scenario:

1 Connect to the Quality Center server (see "Connecting LoadRunner to Quality Center" on page 236).

2 In the Scenario Groups pane, click the **Add Group** button. The Add Group dialog box opens.
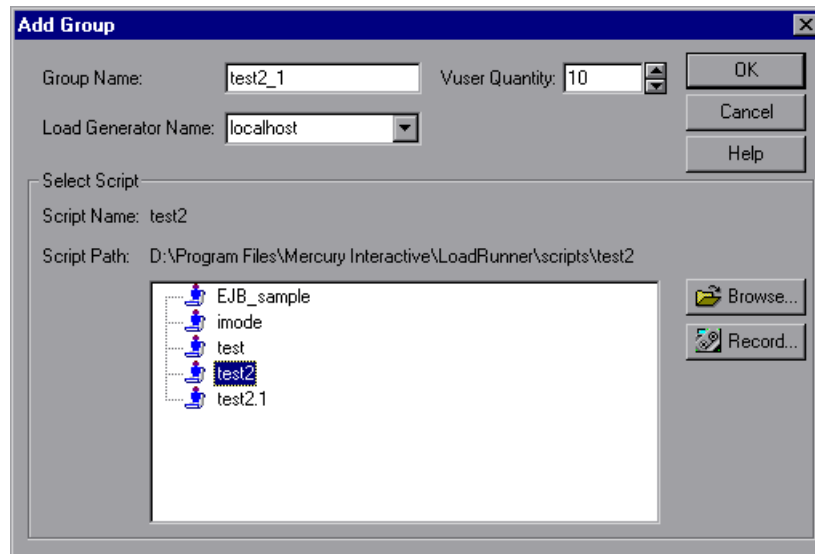
3 Click **Browse**. The Open Test from Quality Center Project dialog box opens and displays the test plan tree.

4 Select the script and click **OK**. The Script Path field displays [TD], the full subject path, and the script name.

For example:

[TD]\Subject\System\test_qc

5 Click **OK** to close the Add Group dialog box. The script is displayed in the Scenario Groups pane.

Adding a Vuser Script to a Goal-Oriented Scenario

When you create a goal-oriented scenario, you use the Add Script dialog box to add scripts.

To add a Vuser script to a goal-oriented scenario:

 1 Connect to the Quality Center server (see "Connecting LoadRunner to Quality Center" on page 236).

 2 On the Scenario Scripts pane toolbar, click the **Add Script** button. The Add Script dialog box opens.
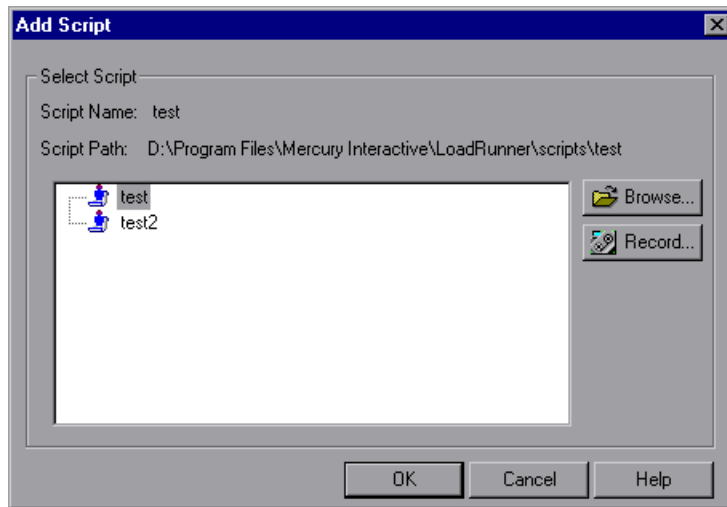
 3 Click **Browse**. The Open Test from Quality Center Project dialog box opens and displays the test plan tree opens.

 4 Select the script and click **OK**. The Script Path field displays [TD], the full subject path, and the script name.

For example:

[TD]\Subject\System\test_qc

 5 Click **OK** to close the Add Script dialog box. The script appears in the Script Path column in the Scenario Scripts pane.

14

Using Functional Testing Scripts in LoadRunner

Using LoadRunner you can test and monitor how your application's *functionality* is affected by heavy load. LoadRunner can integrate functional testing scripts in the form of *GUI Vuser scripts* into a load testing scenario. These GUI Vuser scripts are created using HP Functional Testing software - QuickTest Professional or WinRunner.

This chapter includes:

About Using Functional Testing Scripts in Loadrunner

HP Functional Testing software (QuickTest and WinRunner) enable you to create complex tests that examine the full spectrum of your application's functionality.

LoadRunner can integrate these functional testing scripts into a load testing scenario in the form of GUI Vuser scripts. These scripts, that have already been designed and debugged in QuickTest or WinRunner, can be used as the basis of your load test.

The main advantages of running functional testing scripts in LoadRunner are:

➤ to check how your application's functionality is affected by heavy load.

➤ to measure the response time that a typical user experiences on the client side while your application is under load (end-to-end response time).

For example, you can add QuickTest or WinRunner test scripts to specific points in a LoadRunner scenario to confirm that the application's functionality is not affected by the extra load at those sensitive points.

Another advantage of using a GUI Vuser script as part of your LoadRunner scenario is that the GUI Vuser script runs on your screen during the scenario, enabling you to watch the actual steps executed by the Vuser in real time.

Introducing GUI Vuser Scripts

GUI Vusers enable you to measure and monitor end-to-end user response times while your client/server system is under load. A GUI Vuser emulates the complete environment of a human user.

For example, a human user sits at a machine, operates applications using the keyboard and the mouse, and reads information on the machine's monitor. Similarly, a GUI Vuser runs on its own machine and operates applications. A GUI Vuser can be programmed to read and act on information that appears on its machine's display.

Suppose that you have a bank server that services many automatic teller machines (ATMs). You could create a GUI Vuser script that:

➤ opens the ATM application

➤ enters an account number

➤ enters the amount of cash to be withdrawn

➤ withdraws cash from the account

➤ checks the balance of the account

➤ closes the ATM application

➤ repeats the process

The actions of each GUI Vuser are described in a GUI Vuser script. You use QuickTest and WinRunner to create GUI Vuser scripts.

You monitor and manage GUI Vusers using the LoadRunner Controller. For instance, you can use the Controller to run, pause, or view Vusers, and to monitor scenario status.

Note: You cannot use VuGen to run a GUI Vuser script. You use the Controller to run a GUI Vuser script as part of a scenario; you use WinRunner or QuickTest to run a GUI Vuser script in standalone mode.

Understanding GUI Vuser Technology

GUI Vusers measure real end-to-end response times. End-to-end response times represent the total time that a user waits for a response after submitting a request. End-to-end response times include GUI response times as well as network and server response times.
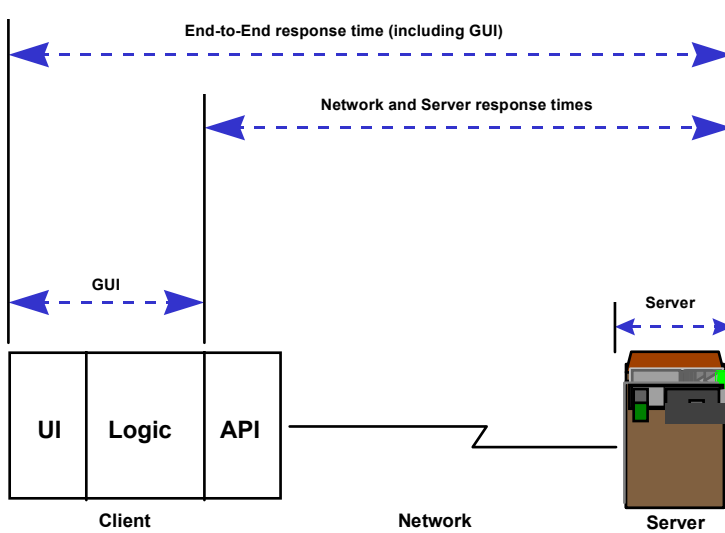
Using QuickTest to Create GUI Vuser Scripts for LoadRunner

When creating test scripts in QuickTest that are going to be used as GUI Vuser scripts in a Loadrunner testing scenario, you need to follow certain guidelines to ensure smooth integration of the script. For detailed explanations about creating tests in QuickTest see the *QuickTest Professional User's Guide*.

This section describes the following general guidelines:

➤ Limitations

➤ Including Transactions

➤ Saving Integration Data with Your Tests

➤ Adding Statements

➤ Designing Tests for LoadRunner

Limitations

QuickTest offers several features that are designed specifically for integration with LoadRunner. Some QuickTest features, however, may not be available when they are integrated with LoadRunner. For more information about specific limitations, see the *QuickTest Professional Readme* section.

Including Transactions

To measure the performance of the server, you define **transactions**. A transaction represents an action or a set of actions that you are interested in measuring. You define transactions within your Vuser script by enclosing the appropriate sections of the script with **start** and **end** transaction statements.

For example, you can define a transaction that measures the time it takes for the server to process a request to view the balance of an account and for the information to be displayed at the ATM.

Note: LoadRunner only provides performance information for data that is included within a transaction. Therefore, your QuickTest test must include transactions to be used by LoadRunner.

You can insert **StartTransaction** and **EndTransaction** statements using the **Insert** > **Start Transaction** and **Insert** > **End Transaction** menu options or tool bar buttons to insert the statement. For more information on these options, see the QuickTest Professional documentation.

Saving Integration Data with Your Tests

To enable integration with LoadRunner Virtual User technology, QuickTest must generate special integration files. By default, the option to generate this data is enabled. However, you, or someone else working on your QuickTest tests may have disabled the option in order to preserve disk space. Before you begin creating tests for use with LoadRunner, enable this option as follows:

1 In QuickTest, select **Tools** > **Options** or click the **Options** toolbar button. The Options dialog box opens and displays the General tab.

2 In the General tab, confirm that **Save data for integrating with Mercury performance testing and application management software** is selected.

3 If you want to integrate a test with LoadRunner that was saved without this option, open and save the test again after selecting this option.

Tip: To check whether a test was saved with integration data, look for a **<testname>.usr** file in your test folder with the same **modified** date as the test.

Adding Statements

You can use the **Services** object and its associated methods to insert statements that are specifically relevant to performance testing. These include **Abort**, **GetEnvironmentAttribute**, **LogMessage**, **SetTransactionStatus**, **ThinkTime**, **UserDataPoint**, **StartTransaction** and **EndTransaction**. For more information on these methods, see the **Utility** section in the *QuickTest Professional Object Model Reference*.

Designing Tests for LoadRunner

Consider the following design guidelines when designing tests for use with LoadRunner:

➤ The QuickTest tests you use with LoadRunner should be simple tests, designed to pinpoint specific operations.

➤ LoadRunner cannot run nested action iterations.

➤ Do not include references to external actions or other external resources, such as an external Data Table file, environment variable file, shared object repositories, and so forth.

➤ Include transactions in your QuickTest test since LoadRunner only provides performance information for data that is included within a transaction.

Using WinRunner to Create GUI Vuser Scripts for LoadRunner

This section outlines how to use WinRunner to create and enhance GUI Vuser scripts that are going to be used in a LoadRunner testing scenario. For detailed explanations about creating tests in WinRunner see the WinRunner documentation.

Note: Some WinRunner features may not be available when they are integrated with LoadRunner.

This section includes:

➤ "Understanding GUI Vuser Scripts" on page 255

➤ "Inserting Transactions and Rendezvous Points" on page 257

➤ "Using Vuser Functions in GUI Vuser Scripts" on page 258

➤ "Sending Messages to the Controller" on page 259

➤ "Obtaining Information about Vusers and Load Generators" on page 260

Understanding GUI Vuser Scripts

WinRunner is a complete development environment for creating, editing, and debugging Windows-based GUI Vuser scripts. Using WinRunner, you record the actions of a human user on an application.

For example, you could record a user entering an account number into an ATM and then withdrawing fifty dollars. These actions are automatically transcribed into a script in HP's Test Script Language (TSL).

GUI Vuser scripts are written in TSL—HP's Test Script Language. TSL is a high-level C-like programming language. It combines the power and flexibility of a conventional programming language with functions designed specifically for testing. For additional information about TSL, see the *TSL Online Reference*.

This section presents a basic Vuser script, created in WinRunner. The script starts an ATM application (mratm.exe), enters an account number, deposits fifty dollars, and then closes the application.

Note: This script will not work on a Unix machine.

The first section of the script starts an application and moves it to a new location on the screen. The **system** function starts the ATM application. The **win_move** function moves the ATM application to a specified location on the screen.

```
# Initialize and invoke ATM client application.
invoke_application("c:\mratm.exe","","",SW_SHOWMINIMIZED);
win_move ("Mercury ATM", 325, 0);
```

Next, the Vuser enters an account number into the ATM. The **set_window** function activates the ATM window. The **edit_set** function instructs the Vuser to enter the account number into the ATM's account field.

```
# Type in account number in the Account field.
account = 100;
set_window ("Mercury ATM");
edit_set ("Account", account);
```

After entering the account number, the Vuser enters the amount it wants to deposit and presses the deposit button. The **edit_set** function enters the amount to be deposited in the amount field. The **button_press** function tells the Vuser to press the ATM's Deposit button.

```
# Enter the amount to be deposited in the amount field.
amount = 50;
set_window ("Mercury ATM");
edit_set ("Amount", amount);
# Press the Deposit button.
button_press ("Deposit");
```

The final section of the test tells the Vuser to close the ATM application. The **menu_select_item** function selects the Exit command from the File menu.

```
# Close client application.
menu_select_item ("File; Exit");
```

Inserting Transactions and Rendezvous Points

To measure the performance of the server, you define **transactions**. A transaction represents an action or a set of actions that you are interested in measuring. You define transactions within your Vuser script by enclosing the appropriate sections of the script with **start** and **end** transaction statements. For example, you can define a transaction that measures the time it takes for the server to process a request to view the balance of an account and for the information to be displayed at the ATM.

Note: LoadRunner only provides performance information for data that is included within a transaction. Therefore, your WinRunner test must include transactions to be used by LoadRunner.

You insert **rendezvous points** into Vuser scripts to emulate heavy user load on the server. **Rendezvous points** instruct Vusers to wait during test execution for multiple Vusers to arrive at a certain point, so that they may simultaneously perform a task. For example, to emulate peak load on the bank server, you can insert a rendezvous point instructing 100 Vusers to deposit cash into their accounts at the same time.

After recording a basic Vuser script using WinRunner, you manually insert:

➤ transaction statements into the script to measure the performance of the server.

➤ rendezvous statements into the script to emulate a specific user load.

To mark the start of a transaction:

Insert a **start_transaction** statement into the Vuser script.

To mark the end of a transaction:

Insert an **end_transaction** statement into the Vuser script. For the syntax of the **start_transaction** and **end_transaction** functions, see the *TSL Online Reference* (available from the WinRunner Help menu).

To insert a rendezvous point:

Insert a **rendezvous** statement into the Vuser script. For the syntax of the **rendezvous** function, see the *TSL Online Reference* (available from the WinRunner Help menu).

Using Vuser Functions in GUI Vuser Scripts

This section lists the Vuser functions that you can use to enhance your GUI Vuser scripts. For syntax and examples of the functions, see the sections that follow or the *TSL Online Reference* (available from the WinRunner Help menu).

Function	Description
declare_rendezvous	Declares a rendezvous.
declare_transaction	Declares a transaction.
end_transaction	Marks the end of a transaction for performance analysis.
error_message	Sends an error message to the Controller.
get_host_name	Returns the name of a load generator.
get_master_host_name	Returns the name of the Controller load generator.
lr_whoami	Returns information about the Vuser executing the script.
output_message	Sends a message to the Controller.
rendezvous	Sets a rendezvous point in a Vuser script.
start_transaction	Marks the beginning of a transaction for performance analysis.
user_data_point	Records a user-defined data sample.

Sending Messages to the Controller

When you run a scenario, the Controller's Output window displays valuable information about script execution. In addition to the messages automatically sent by WinRunner, you can include statements in each script that send error and notification messages to the Controller. For example, you could insert a message that displays the current state of an application. After scenario execution, you can save these messages to a file.

The **error_message** function sends an error message to the Controller's Output window. The syntax of this function is:

error_message (*message*);

where *message* is a text string.

In the following example, the Vuser script sends a message when a fatal error occurs during script execution.

```
if (fatal_error < 0){
 mess="fatal error - Exiting.";
 error_message (mess);
 texit (1);
 }
```

The **output_message** function is used to send a special notification that is not an error message. The syntax of this function is:

output_message (*message***);**

where message is a text string.

For further information on the **error_message** and the **output_message** functions, see the *TSL Online Reference* (available from the WinRunner Help menu).

Obtaining Information about Vusers and Load Generators

During scenario execution, you can obtain the identity of:

➤ the Vusers performing a task at a particular moment in the scenario

➤ the load generator executing a script

➤ the machine running the Controller

For example, you could program statements into a Vuser script to return the ID of each active Vuser currently using an application, and print this information to a file.

The following functions obtain information about Vusers and load generators:

Function	Description
lr_whoami	Returns the name of a Vuser and the Vuser group it belongs to.
get_host_name	Returns the name of the machine executing the script.
get_master_host_name	Returns the name of the machine running the Controller.

In the following example, the **get_host_name** function returns the name of the load generator currently running the script. The **print** statement saves the information to a file.

```
my_host_name = get_host_name();
print("my local load generator name is:" & my_host_name) > vuser_file;
```

For more information about these functions, see the *TSL Online Reference* (available from the WinRunner Help menu).

After recording a basic Vuser script, you insert statements into the script that measure the performance of the server (transactions), and create a synchronized user load (rendezvous points). For more details about GUI Vusers, see your *HP LoadRunner Online Function Reference*.

Running GUI Vuser Scripts in a LoadRunner Scenario

After you have created your GUI Vuser script in QuickTest or WinRunner, you are ready to integrate the script into a LoadRunner scenario.Before you run the LoadRunner scenario you should consider the following guidelines:

➤ You can run only one GUI Vuser concurrently per machine.

➤ Ensure that QuickTest and WinRunner are closed before running the scenario.

➤ In the Run-time Settings for script dialog box, only the **General** categories and sub-categories (**General**, **Iterations**, **Miscellaneous**, **Think Time**) are relevant for QuickTest and WinRunner tests. The **Replay** options are not relevant.

To add a GUI Vuser script to a LoadRunner scenario:

 1 Navigate to the folder containing the test script

➤ For a new scenario, click **Browse** in the New Scenario dialog box.

➤ When adding the script to an existing scenario click **Browse** in the Add Group/Add Script dialog box.

The Open Test dialog box opens.
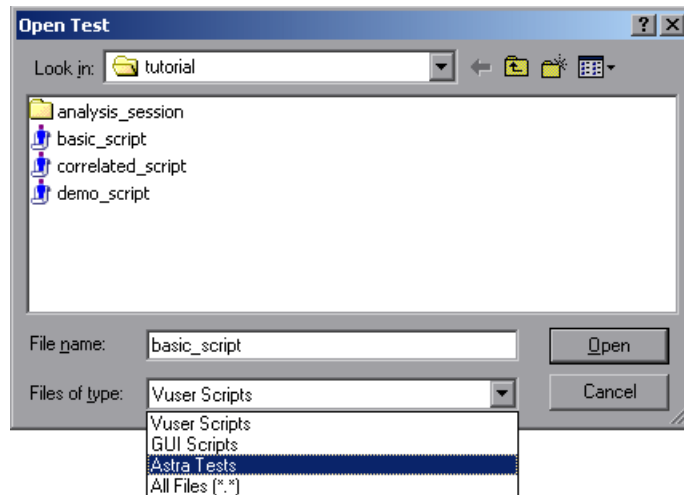
2 In the **Files of Type** box select the appropriate file type.

➤ For WinRunner, select **GUI Scripts.**

➤ For QuickTest, select **Astra Tests.**

3 Navigate to the appropriate script and add it to your scenario.

Part III

Executing a Scenario

15

Running a Scenario

When you run a scenario, LoadRunner generates load on the application you are testing, and measures the system's performance.

This chapter includes:

➤ About Running a Scenario on page 265

➤ Running an Entire Scenario on page 267

➤ Controlling Vuser Groups on page 268

➤ Controlling Individual Vusers on page 275

➤ Manually Releasing Vusers from a Rendezvous on page 277

➤ Manually Adding Vusers to a Running Scenario on page 278

About Running a Scenario

When you run a scenario, the Vuser groups are assigned to their load generators and execute their Vuser scripts. During scenario execution, LoadRunner:

➤ records the durations of the transactions you defined in the Vuser scripts

➤ performs the rendezvous included in the Vuser scripts

➤ collects error, warning, and notification messages generated by the Vusers

You can run an entire scenario unattended, or you can interactively select the Vuser groups and Vusers that you want to run. When the scenario starts running, the Controller first checks the scenario configuration information. Next, it invokes the applications that you selected to run with the scenario. Then, it distributes each Vuser script to its designated load generator. When the Vuser groups are ready, they start executing their scripts.

While the scenario runs, you can monitor each Vuser, view the error, warning, and notification messages generated by the Vusers, and stop both Vuser groups and individual Vusers. You can instruct LoadRunner to allow an individual Vuser or the Vusers in a group to complete the iterations they are running before stopping, to complete the actions they are running before stopping, or to stop running immediately. For more information, see "Configuring Scenario Run-Time Settings" on page 218.

Note: When automatically stopping Vusers in a goal-oriented scenario, LoadRunner stops running the Vusers immediately.

You can also activate additional Vusers while the scenario is running, using the Run/Stop Vusers dialog box. For more information, see "Manually Adding Vusers to a Running Scenario" on page 278.

The scenario ends when all the Vusers have completed their scripts, when the duration runs out, or when you terminate it.

The following procedure outlines how to run a scenario:

1 Open an existing scenario or create a new one.

2 Configure and schedule the scenario.

3 Set the results directory.

4 Run and monitor the scenario.

Running an Entire Scenario

You can run all the Vusers and Vuser groups in a scenario, or you can select the specific Vuser groups and Vusers that you want to run. When you run an entire scenario, LoadRunner does not begin running Vusers until they all have reached the **Ready** state. However, if you run individual groups or Vusers, LoadRunner runs each Vuser as soon as it reaches the **Ready** state.

The following section describes how to run an entire scenario. "Controlling Vuser Groups" on page 268, and "Controlling Individual Vusers" on page 275 describes how to manipulate Vuser groups and individual Vusers.

To run an entire scenario:

 1 Open an existing scenario or create a new one. Select the **Run** tab. The Scenario Groups pane appears in the top left corner of the screen.
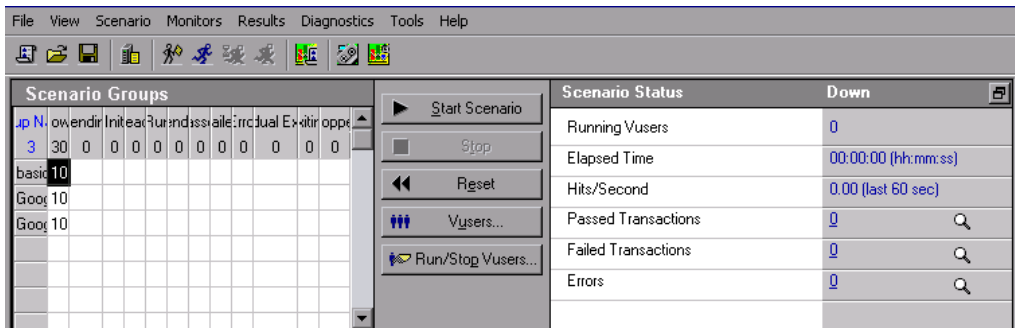
 2 Select **Scenario** > **Start**, or click **Start Scenario**. The Controller starts initializing the Vusers and distributing them to their designated load generators, where they begin to execute their Vuser scripts.

Note: The Controller begins running the scenario according to the start time set in the Scenario Start dialog box.

If you have not specified a results directory for the scenario, the Set Results Directory dialog box opens.

During scenario execution, you can manipulate individual Vusers and Vuser groups. This is described in "Controlling Vuser Groups" below and "Controlling Individual Vusers" on page 275.

3 Select **Scenario** > **Stop**, or click **Stop** to terminate the scenario. If you selected the **Stop immediately** option in the Run-Time Settings tab of the Options dialog box, all of the Vusers in the scenario move to the **Exiting** status.

If you selected the **Wait for the current iteration to end before exiting** or **Wait for the current action to end before exiting** options in the Run-Time Settings tab of the Options dialog box, the Vusers in the scenario move to the **Gradual Exiting** status and exit the scenario gradually. To stop the Vusers immediately, click **Stop Now**.

4 Select **Scenario** > **Reset**, or click **Reset** to reset all Vusers to their pre-scenario **Down** status.

Controlling Vuser Groups

You can run an entire scenario as described above, or you can manipulate individual Vuser groups in the scenario. This section describes how to initialize, run, pause, stop and reset Vuser groups.

Initializing Vuser Groups

Initializing a Vuser group distributes the Vusers in the group to their designated load generators so that they are ready to execute their scripts. By initializing all of the Vusers in a group before running them, you can ensure that they all begin executing the scenario at the same time.

To initialize a Vuser group:

1 Select the Vuser group or groups that you want to initialize.

2 Click **Initialize Vusers**, or right-click the Vuser group or groups that you want to initialize and select **Initialize Vusers**. The Vuser group's status changes from **Down** to **Pending** to **Initializing** to **Ready**. If a Vuser group fails to initialize, the Vuser group status changes to **Error**.

Running Vuser Groups

Running a Vuser group tells the Vuser group to execute its script.

To run a Vuser group:

1 Select the Vuser group or groups that you want to run.

2 Click **Run Vusers**, or right-click the Vuser group or groups that you want to run and select **Run Vusers Until Complete**. The Vuser groups execute their scripts. If you run a Vuser group in the **Down** or **Error** state, LoadRunner initializes and then runs the Vuser group.

Note: You can instruct LoadRunner to randomly run only one Vuser in a group by right-clicking the Vuser group and selecting **Run One Vuser Until Complete**. A Vuser script log opens, displaying run-time information about the Vuser. For more information about the Vuser log, see "Viewing the Vuser Script Log" on page 295.

Pausing Vuser Groups

Pausing a Vuser group temporarily stops script execution. The Pause command changes the status of a Vuser group from **Running** to **Paused**.

Note: Pausing a Vuser group will affect its transaction response time.

To pause a Vuser:

1 Select the Vuser group or groups that you want to pause.

2 Select **Pause Vusers** from the right-click menu. The Vusers groups temporarily stop script execution.

Stopping Vuser Groups

Stopping a Vuser group stops script execution. If you stop a Vuser group, the group still appears in the Vuser group list.

To stop a Vuser group:

1 Select the Vuser group or groups that you want to stop.

2 Click **Stop Vusers**, or right-click the Vuser group or groups and select **Stop**. The Vuser groups stop executing their scripts immediately.

If you selected the **Wait for the current iteration to end before exiting** or **Wait for the current action to end before exiting** options in the Run-Time Settings tab of the Options dialog box and want to gradually stop a Vuser group in the **Run** state, click the **Gradual Stop** button, or right-click the Vuser group and select **Gradual Stop**. The Vusers in the group move to the **Gradual Exiting** status and exit the scenario gradually.

Note: If the Vusers are not in the **Run** state, the Gradual Stop option is disabled.

Resetting Vuser Groups

Resetting causes all of the Vusers in a group to revert to their pre-scenario **Down** status.

To reset a Vuser group:

1 Select the Vuser group or groups that you want to stop.

2 Right-click the Vuser group or groups that you want to stop, and select **Reset Vusers**. The Vuser groups revert to their pre-scenario **Down** status.

Understanding the Run View Tab

The Run tab displays the Scenario Groups window, a scenario status summary, and graphs with online information generated during script execution.

➤ **Scenario Groups pane.** Displays each Vuser group and its current status.

You can perform the following actions on individual Vuser groups in the scenario by right-clicking a group in the Scenario Groups window:

➤ **Reset IDs.** Resets the IDs of the Vusers in the group.

➤ **Initialize Vusers.** Distributes the Vusers in the group to their designated load generators so that they are ready to execute their script. The Vuser group's status changes from **Down** to **Pending** to **Initializing** to **Ready**. If a Vuser group fails to initialize, the Vuser group status changes to **Error**.

By initializing all of the Vusers in a group before running them, you can ensure that they all begin executing the scenario at the same time.

➤ **Run Vusers Until Complete**. The selected Vusers until complete. If you run Vusers in the **Down** or **Error** state, LoadRunner initializes and then runs the Vusers.

➤ **Run One Vuser Until Complete.** Instructs the Controller to randomly run one of the Vusers in the Vuser group until it completes running. The Vuser Log opens, displaying run-time information about the Vuser.

➤ **Pause Vusers.** Temporarily pauses running the Vusers group. The status of the Vuser group changes from **Running** to **Paused**.

Note: Pausing a Vuser group affects its transaction response time.

➤ **Gradual Stop.** Instructs the Controller to complete the current iteration or action before stopping the Vuser group.

Note: This option is only available when the Vuser group is in the **Run** state, if you selected the **Wait for the current iteration to end before exiting** or **Wait for the current action to end before exiting** options in the Run-Time Settings tab of the Options dialog box.

➤ **Stop.** Instructs the Controller to stop running the Vuser group immediately.

➤ **Reset Vusers.** Resets all Vusers in the group to their pre-scenario **Down** status.

➤ **Enable.** Enables the Vuser group to participate in the scenario.

➤ **Disable.** Disables the Vuser group so that it no longer participates in the scenario.

➤ **Show Vuser.** Opens a Run-Time Viewer for each Vuser in the group.

➤ **Hide Vuser.** Closes the Run-Time Viewers that were opened.

➤ **Show Vuser Log.** Opens a script log containing run-time information for each Vuser in the group. The Vuser script log is refreshed, by default, every 1000 milliseconds.

➤ **Hide Vuser Log.** Closes the Vuser script logs.

➤ **Sort By Name.** Sorts the groups alphabetically, by name.

➤ **Start Scenario.** Instructs the Controller to start initializing the Vusers in the scenario and distributing them to their designated load generators, where they begin to execute their Vuser scripts.

Note: The Controller begins running the scenario at the time stipulated in the Scenario Start dialog box.

We do not recommend changing the Time/Date and Time Zone settings on the Controller and load generators during the load test run.

➤ **Stop.** Instructs the Controller to stop running the scenario. If you selected the **Stop immediately** option in the Run-Time Settings tab of the Options dialog box, all of the Vusers in the scenario move to the **Exiting** status. If you selected the **Wait for the current iteration to end before exiting** or **Wait for the current action to end before exiting** options in the Run-Time Settings tab of the Options dialog box, the Vusers in the scenario move to the **Gradual Exiting** status and exit the scenario gradually.

➤ **Stop Now.** Instructs the Controller to stop running the scenario immediately.

➤ **Reset.** Resets all the Vuser groups in the scenario to their pre-scenario DOWN status.

➤ **Vusers.** Opens the Vusers dialog box, enabling you to view the status of each of the Vusers in a Vuser group.

➤ **Run/Stop Vusers.** Opens the Run/Stop Vusers dialog box, enabling you to activate additional Vusers.

➤ **Pause/Resume Scheduler.** Pauses or resumes the scenario schedule. (Displayed during scenario execution only.)

➤ **Scenario Status Window.** Displays a synopsis of the running scenario. You can view details of individual transactions and errors by clicking the magnifying glass icon.

Note: You can detach the Scenario Status window from the Run view by clicking the button in the upper-right corner. This enlarges the Scenario Groups window.

➤ **Graphs.** To view a list of the available graphs, select **View** > **Show Available Graphs**. To hide the graph tree view, select **View** > **Hide Available Graphs**, or click the **X** button in the top right corner of the Available Graphs list.

To display a graph, click it in the left pane and drag it to the right pane. By default, four graphs are displayed. To customize your online graph display, select **View** > **View Graphs** and select the number of graphs you want to view. You can view up to 16 graphs simultaneously. To display only one graph, double-click the graph in the right pane. To return to the previous view, double-click the graph again. Below the graphs, a legend displays the statistics of the selected graph: color, scale, measurement/status, machine, maximum, minimum, and average values, standard deviation, and last.

You can perform the following actions on a graph by using the Monitors menu or right-clicking the graph:

➤ Graph configuration

➤ Graph duplication

➤ Freeze/Release graph

➤ Export graph to HTML

➤ Overlay graphs

You can perform the following actions on a measurement by right-clicking it:

➤ Add/Delete measurement

➤ Show/Hide measurement

➤ Measurement configuration

➤ Measurement description

Controlling Individual Vusers

You can also manipulate individual Vusers within the Vuser groups you have defined. This section describes how to initialize, run, and stop individual Vusers.

To control an individual Vuser:

1 Select a Vuser group and click the **Vusers** button. The Vusers dialog box opens with a list of the ID, Status, Script, Load Generator, and Elapsed Time (since the beginning of the scenario) for each of the Vusers in the group.
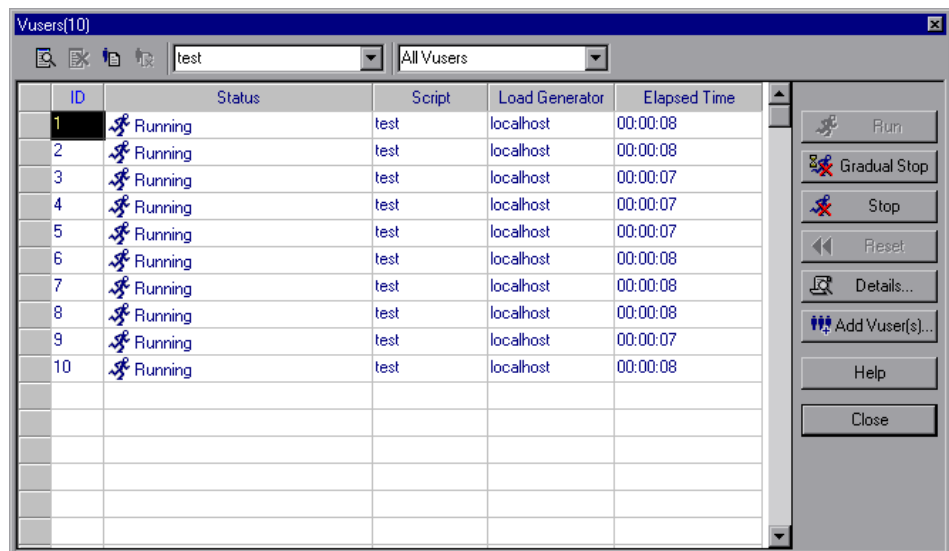
You can control an individual Vuser using the following utilities:

➤ Select a Vuser and click **Run** to run it.

➤ Select a Vuser and click **Stop** to stop it immediately from running.

➤ If you selected the **Wait for the current iteration to end before exiting** or **Wait for the current action to end before exiting** options in the Run-Time Settings tab of the Options dialog box, and want to gradually stop a Vuser in the RUN state, click the **Gradual Stop** button. The Vuser moves to the **Gradual Exiting** status and exits the scenario gradually.

➤ To pause a Vuser, right-click it and select **Pause**. Pausing a Vuser will affect its transaction response time.

➤ To revert a Vuser's status to DOWN, select it and click **Reset**.

➤ To initialize a Vuser, right-click it and select **Initialize Vuser/s**.

➤ To reset the IDs of the Vusers in a group, right-click the Vusers and select **Reset IDs**.

➤ To filter the Vusers listed, right-click in one of the columns and select **Filter Vusers.** Select the way in which you want to filter the Vusers. Alternatively, you can select the filter option you want to use from the filter selector in the upper-right corner of the Vusers dialog box.

➤ To sort the Vusers listed, right-click in one of the columns and select **Sort Vusers**. Select the way in which you want to sort the Vusers.

➤ To view a Vuser executing its assigned script, select the Vuser and click the **Show** button. The Run-Time Viewer opens and displays snapshots of the pages returned to a Vuser, allowing you to see the Vuser executing the script. The Run-Time Viewer does not function as a browser, so the displayed images are snapshots and do not present all aspects of the replay.

➤ The **Options** menu items allow you to select the types of controls to display.

➤ The **View** menu items allow you to open various toolbars and views.

➤ To close the Run-Time Viewer, click the **Hide** button.

➤ To view the Vuser script log, click the **Vuser log** button. A script log, such as the following, appears.
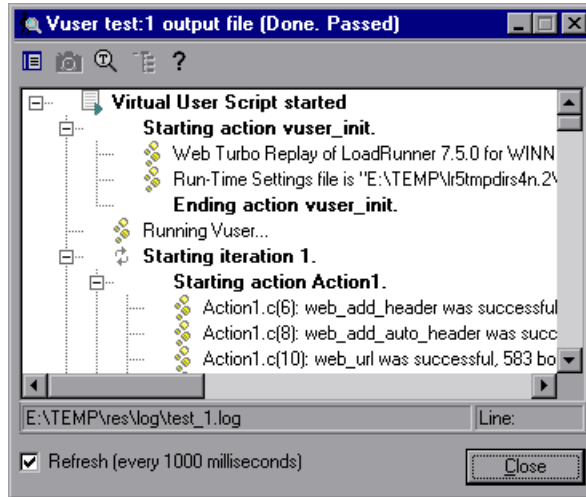
➤ To close the Vuser script log, click the **Close** button. For more information on the Vuser script log, see page 295.

2 Click **Close** to close the Vusers dialog box.

Manually Releasing Vusers from a Rendezvous

While you run a scenario, you can manually release Vusers from a rendezvous before the Controller releases them.

To manually release Vusers from a rendezvous:

1 Select **Scenario** > **Rendezvous**. The Rendezvous Information dialog box opens.

2 Select a rendezvous from the Rendezvous list.

3 Click **Release**. The Vusers in the rendezvous are released.

Manually Adding Vusers to a Running Scenario

During a running scenario, you can manually control the addition of new Vusers using the Run/Stop Vusers dialog box. The dialog box differs depending on the scenario mode you are running:

➤ If you are running a scenario in the Vuser Group Mode, you control the number of new Vusers that can be added to each Vuser Group, and the load generators on which these additional Vusers will run.

➤ If you are running a scenario in the Percentage Mode, you control the number of new Vusers that can be distributed among the Vuser scripts according to the percentage you define, and the load generators on which these additional Vusers will run.

Note: If you are running a scenario or Vuser group using Schedule Builder settings, these settings will be applied to all Vusers that are manually added to the scenario or Vuser group during the scenario run. For more information, see "Adding Vusers to a Scheduled Scenario During a Run" on page 163.

To add Vusers to a running scenario:

1 Select **Scenario** > **Run/Stop Vusers**, or click **Run/Stop Vusers** in the Scenario Groups pane of the Run view. The Run/Stop Vusers dialog box opens. If you are in the Vuser Group Mode, the dialog box displays the Vuser groups included in the scenario.
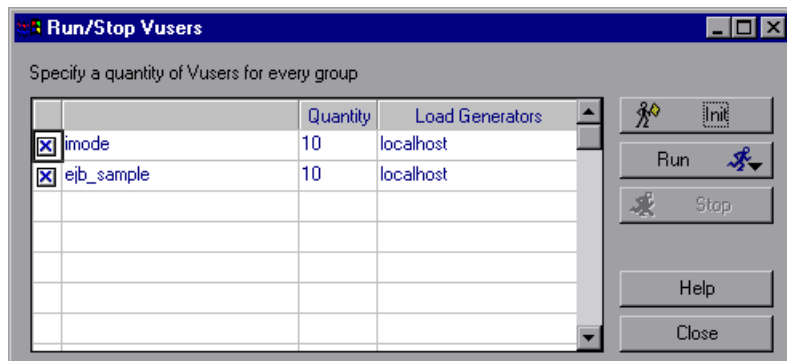
If you are in the Percentage Mode, the Run/Stop Vusers dialog box displays the Vuser scripts included in the scenario.
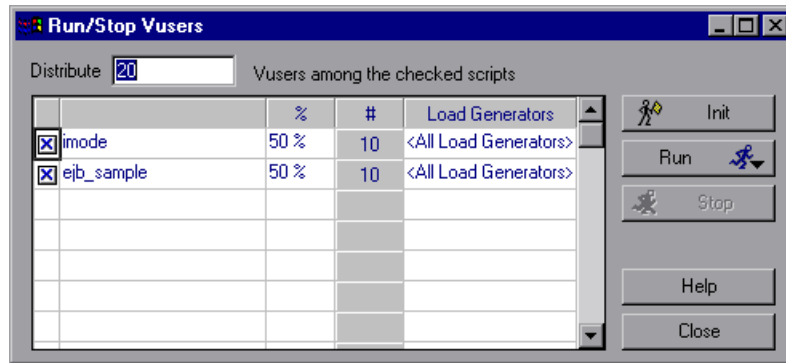
2 If you are in the Vuser Group Mode, enter the number of Vusers you want to run for each group in the Quantity column.

If you are in the Percentage Mode, enter the number of Vusers and the percentage in which you want these Vusers to be distributed among the checked Vuser scripts. LoadRunner automatically distributes the number of Vusers you entered.

3 To disable a Vuser group/script, clear the check box to the left of the group/script name. A group/script will automatically appear disabled if it is disabled in the Design view.

Note: If you disable a Vuser group in the Vuser Group Mode, no Vusers will be distributed to it. If you disable a Vuser script in the Percentage Mode, no Vusers will be distributed to it, and the unused percentage of the Vusers will not be distributed among the remaining scripts, unless you define a zero percent value for the disabled script.

---

4 To change the load generator on which the Vuser group/script will run, select a different load generator from the Load Generator column.

To use a load generator that does not appear, select **Add** from the Load Generator Name list. The Add New Load Generator dialog box opens. Add the name, platform, and temporary directory of the new load generator and click **OK**.
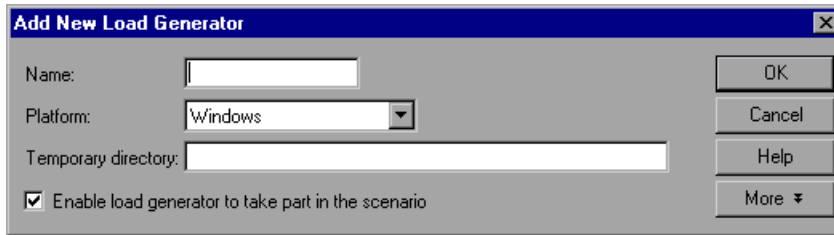
If you are in the Percentage Mode, you can select more than one load generator to run the Vuser script. From the Load Generator Name list, select the load generators and click **OK**. To use all the load generators in the list, click the **All Generators** button.

Note: If you defined more than one load generator for a script, the added Vusers are proportionally distributed among the defined load generators.

---

5 Click **Init** to initialize the number of Vusers you added.

6 Click **Run**, and select a run option.

7 Click **Stop** to stop the Vusers that are running on the load generators defined in the Run/Stop Vusers dialog box.

8 Click **Close** to close the Run/Stop Vusers dialog box.

Example of a Manually Controlled Scenario

The example below shows the Run/Stop Vusers dialog box from a scenario running in the Percentage Mode.
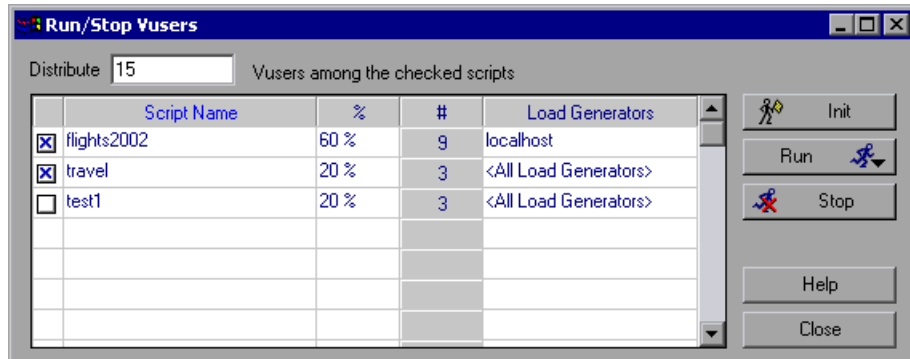
The number of Vusers that are distributed among the checked scripts is 15. The % column indicates that 60% of these Vusers are distributed to the script **flights2002**, and 20% to both **travel** and **test1**. In accordance with these percentages, the # column indicates that nine Vusers are distributed to **flights2002** and three to **travel** and **test1**.

Note: The unused percentage of the Vusers from the disabled **test1** script are not distributed among the remaining scripts, because a percentage value has been defined for this script.

When an action (**Init**, **Run**, **Stop**) is selected from the Run/Stop Vusers dialog box, the Controller runs only the number of Vusers that appear in the # column. In this example, nine Vuser are initialized, run, or stopped in the **flights2002** script, and three in the **travel** script.

All Vusers distributed to the **flights2002** script are run on the **localhost** load
generator. For the **travel** script, Vusers are proportionally distributed among
all defined load generators.

Note: Load generator balancing is applied to a manually controlled scenario
in the Percentage Mode where there are other load generators assigned to
Vuser scripts. For more information, see "Load Balancing" on page 193.

Understanding the Run/Stop Vusers Dialog Box

The Run/Stop Vusers dialog box lets you manually activate additional
Vusers. This dialog box differs depending on whether you are running a
scenario in the Vuser Group Mode or in the Percentage Mode.

➤ **Specify a quantity of Vusers for every group.** Enter the number of Vusers
you want to run for each group in the Quantity column "#" (Vuser Group
Mode).

➤ **Distribute X Vusers among the checked scripts.** Enter the number of Vusers
you want to distribute by percentage among the checked Vuser scripts.
LoadRunner automatically distributes the number of Vusers you entered
(Percentage Mode).

➤ **Run/Stop Vusers table.**

  ➤ **Group/Script Name.** Displays the names of the Vuser groups or scripts
  running in the scenario.

  ➤ **%.** Indicates the percentage of Vusers distributed to each Vuser script
  (Percentage mode only).

  ➤ **#.** Indicates the number of Vusers distributed to each Vuser script.

  ➤ **Load Generators.** Indicates the load generators on which the Vusers will
  be run. If more than one load generator is defined for a script, the added
  Vusers are proportionally distributed among the defined load generators.

Note: To disable a Vuser group/script, clear the check box to the left of the group/script name. A group/script will automatically appear disabled if it is disabled in the Design view.

If you disable a Vuser script, no Vusers will be distributed to it. However, 100 percent of the Vusers will not be distributed among the remaining scripts, unless you define a 0 percent value for the disabled script.

➤ **Init.** Distributes the Vusers that you added to their designated load generators so that they are ready to execute their scripts. The Controller first initializes the Vusers in your scenario that have not yet been run, on the load generators defined in the current dialog box. It then adds additional Vusers, as required, to reach the quantity defined in the current dialog box.

➤ **Run.** Select one of the following options:

  ➤ **Run Initialized.** Runs the Vusers in the scenario that have already been initialized on the load generators defined in the current dialog box. The Controller runs only those Vusers that have already been initialized, regardless of their quantity.

  ➤ **Run New.** Runs the number of Vusers you specified. The Controller first runs the Vusers in your scenario that have not yet been run, on the load generators defined in the current dialog box. It then adds additional Vusers, as required, to reach the quantity defined in the current dialog box.

➤ **Stop.** Stops the Vusers that are running on the load generators defined in the current dialog box. The Controller stops the Vusers according to the settings you defined in the "Configuring Scenario Run-Time Settings" on page 218.

16

Viewing Vusers During Execution

During scenario execution, you can view the actions that are performed by Vusers.

This chapter includes:

About Viewing Vusers During Execution

LoadRunner lets you view Vuser activity during a scenario.

➤ On the Controller load generators, you can view the Output window, monitor Vuser performance online, and check the status of Vusers executing the scenario.

➤ On remote machines, you can view the Agent summary with information about the active Vusers.

Monitoring Vuser Status

During scenario execution, you can use the Scenario Groups pane in the Run view to monitor the actions of all the Vusers and Vuser groups in the scenario.

The Status field of each Vuser group displays the current state of each Vuser in the group. The following table describes the possible Vuser statuses during a scenario:

Status	Description
Down	The Vuser is down.
Pending	The Vuser is ready to be initialized and is waiting for an available load generator, or is transferring files to the load generator. The Vuser will run when the conditions set in its scheduling attributes are met.
Initializing	The Vuser is being initialized on the remote machine.
Ready	The Vuser already performed the init section of the script and is ready to run.
Running	The Vuser is running. The Vuser script is being executed on a load generator.
Rendezvous	The Vuser has arrived at the rendezvous and is waiting to be released by LoadRunner.
Done.Passed	The Vuser has finished running. The script passed.
Done.Failed	The Vuser has finished running. The script failed.
Error	A problem occurred with the Vuser. Check the Status field on the Vuser dialog box or the output window for a complete explanation of the error.
Gradual Exiting	The Vuser is completing the iteration or action it is running (as defined in **Tools** > **Options** > **Run-Time Settings**) before exiting.
Exiting	The Vuser has finished running or has been stopped, and is now exiting.
Stopped	The Vuser stopped when the Stop command was invoked.

You can also view a synopsis of the running scenario in the box in the upper-right corner of the Run view.

Scenario Status	Running	
Running Vusers	10	
Elapsed Time	00:00:38 (hh:mm:ss)	
Hits/Second	1.48 (last 60 sec)	
Passed Transactions	10	🔍
Failed Transactions	0	🔍
Errors	0	🔍

Note: You can detach the Scenario Status window from the Run view by clicking the button in the upper-right corner. This enlarges the Scenario Groups pane.

The following table describes the runtime information that can be viewed during a scenario:

Scenario Status	Description
Scenario Status	Indicates whether the scenario is RUNNING or DOWN
Running Vusers	Indicates how many Vusers are being executed on a load generator
Elapsed Time	Indicates how much time has elapsed since the beginning of the scenario
Hits/Second	Indicates how many hits (HTTP requests) there have been to the Web site being tested per second that each Vuser has been running
Passed Transactions	Indicates how many transactions have been executed successfully

Scenario Status	Description
Failed Transactions	Indicates how many transactions have been executed unsuccessfully
Errors	Indicates how many problems have occurred with the Vusers

Transactions

You can view details of individual transactions in the Transactions dialog box. To open the Transactions dialog box, in the Scenario Status window click the **Show Snapshot** button to the right of **Passed Transactions** or **Failed Transactions**.

Detail	Description
Name	Lists the names of the individual transactions in a script.
TPS	Displays the number of transactions per second.
Passed	Lists the number of transactions that passed.

Detail	Description
Failed	Lists the number of transactions that failed.
Stopped	Lists the number of transactions that stopped

Viewing the Output Window

While the scenario runs, the Vusers and load generators send error, notification, warning, debug, and batch messages to the Controller. You can view these messages in the Output window.

LoadRunner clears the messages in the Output window at the start of each scenario execution. If you reset a scenario, messages remain in the Output window unless you instruct LoadRunner to delete messages from the Output window upon reset. For more information, see "Options - Output Settings" on page 456.

To view messages in the Output window:

 1 Choose **View** > **Show Output** or click the **Show Snapshot** button to the right of the Errors listing. The Output window opens, displaying error information.
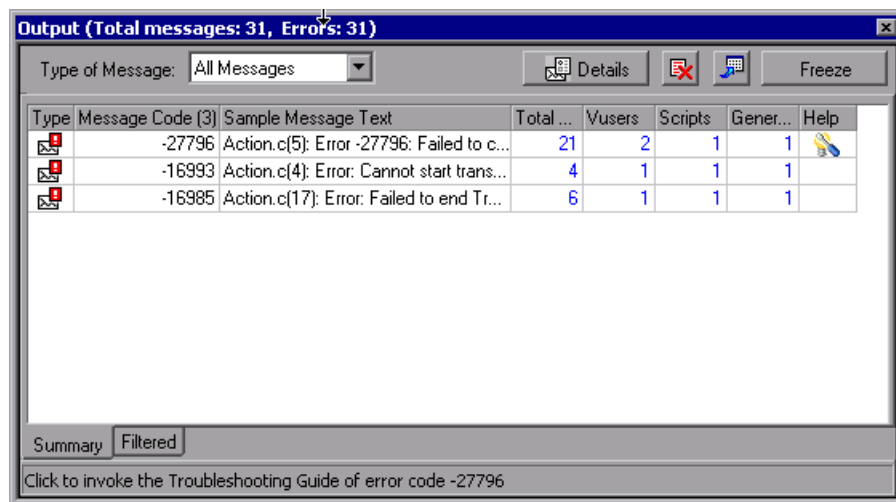
2 In the **Type of Message** box, select a message type to filter.

3 To view a message in detail, select the message and click **Details**. The Detailed Message Text box opens in the Output window displaying the complete sample message text.
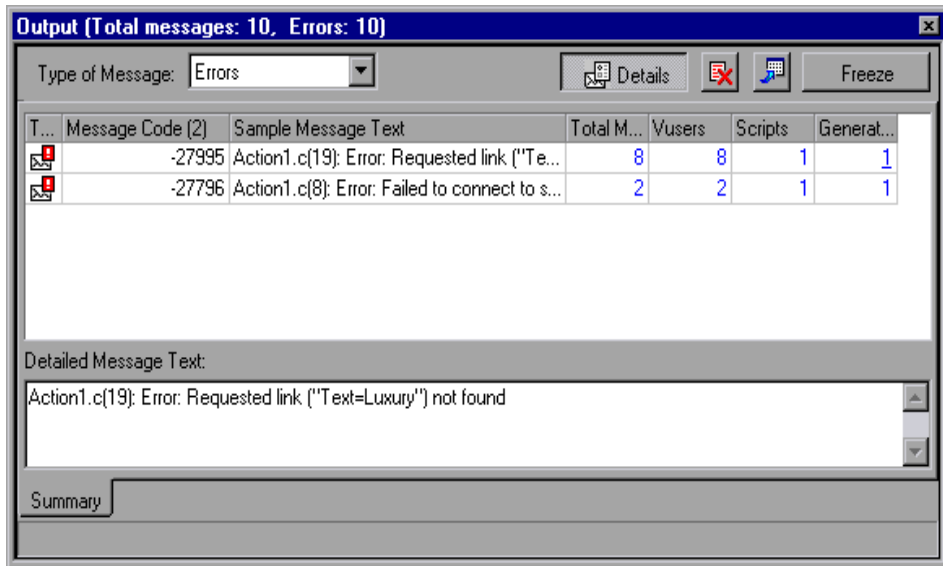
4 To view details of the log information by message, Vuser, script, or load generator, click the blue link in the respective column.

5 To troubleshoot the error, click the link in the Help column.

For more information, see "Viewing Log Information Details" below.

Viewing Log Information Details

You can view details of each message, Vuser, script, and load generator associated with an error code by clicking the blue link in the respective column. The Output window displays a drilled-down view by Vuser, message, script, or load generator in the Filtered tab.

For example, if you drill down on the Vusers column, the Output window displays all the messages with the code you selected, grouped by the Vusers that sent the messages.
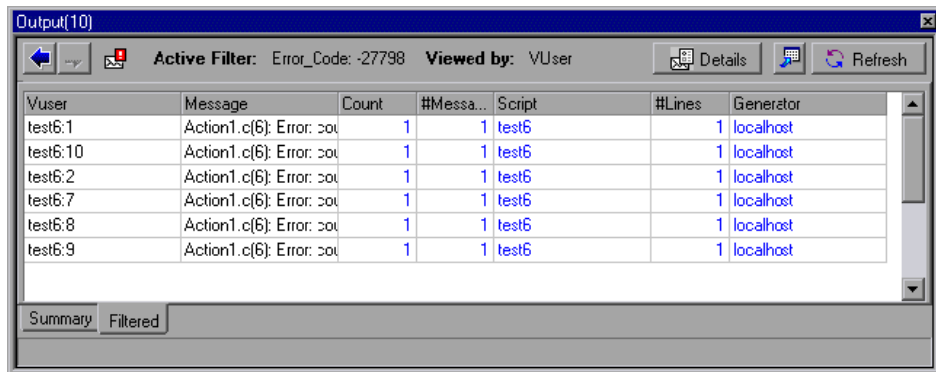
The message type, the message code, and the column that you selected to drill down on, are displayed above the grid.

You can drill down further on the entries displayed in blue. When you drill down on a Vuser, the Vuser log opens. When you drill down on a load generator, the Load Generators dialog box opens, displaying the load generator you selected. When you drill down on a script (or Action or Line Number), VuGen opens, displaying the script you selected.

Note: To limit the number of rows displayed when you drill down, open the **wlrun7.ini** file in any text editor, and located the following line:

MaxOutputUIRowsToShow=0

Change the 0 (no limit) to the number of rows you want to view.

When new messages arrive in the Output window, the Refresh button is enabled. Click **Refresh** to add the new log information to the Filtered tab view.

To move between the various drill down levels, click the **Previous view** and **Next view** buttons in the upper-left corner of the Output window.

Understanding the Output Window

The Output window displays information about the error, notification, warning, debug, and batch messages that Vusers and load generators send to the Controller during scenario execution. The total number of messages sent is displayed in the title bar.

Note: You can also specify the maximum number of Vuser logs that may be displayed simultaneously on the Controller machine. For more information, see Appendix C, "Working in Expert Mode."

Summary Tab

The Summary tab displays summary information about the messages sent during scenario execution. To view details of each message, Vuser, script, and load generator associated with an error code, click the blue link in the respective column.

➤ **Type of Message.** Filters the output messages to display only certain message types. Select one of the following filters:

Icon	Message Type	Description
	All Messages	Displays all message types.
✉	Notifications	Provides run-time information, such as messages sent using **lr_output_message**.
✉	Errors	Usually indicate that the script failed.
✉	Warnings	Indicates that the Vuser encountered a problem, but test execution continued.
✉	Debug	Will be sent only if you enable the debugging feature in Controller (**Tools** > **Options** > **Debug Information**). See "Options - Debug Information Settings" on page 454 for more information.
✉	Batch	Sent instead of message boxes appearing in the Controller, if you are using automation.

➤ **Details.** Displays the full text of the selected output message in the Output window.

➤ **Remove all messages.** Clears all log information from the Output window.

➤ **Export the view:** Saves the Output view to a specified file.

➤ **Freeze/Resume.** Stops updating the Output window with messages. To instruct LoadRunner to resume updating the Output window, click **Resume**. The newly updated log information is displayed in a red frame.

➤ **Message Code.** Displays the code assigned to all similar messages. The number in parentheses indicates the number of different codes displayed in the Output window.

➤ **Sample Message Text.** Displays an example of the text of a message with the specified code.

➤ **Total Messages.** Displays the total number of sent messages with the specified code.

> ➤ **Vusers.** Displays the number of Vusers that generated messages with the specified code.

> ➤ **Scripts.** Displays the number of scripts whose execution caused messages with the specified code to be generated.

> ➤ **Generators.** Displays the number of load generators from which messages with the specified code were generated.

> ➤ **Help.** Displays an icon if there is a link to troubleshooting for this message.

Note: To sort the log information, click the appropriate column header. The messages are sorted in descending/ascending order.

Filtered Tab

The Filtered tab displays a drilled down view by Vuser, message, script, or load generator. For example, if you drill down on the Vusers column, the Filtered tab displays all the messages with the code you selected, grouped by the Vusers that sent the messages.

> ➤ **Previous view/Next view.** Enables you to move between the various drill down levels.

> ➤ **<Type of Message> Icon.** Displays an icon indicating the type of message by which the current Output view is filtered.

> ➤ **Active Filter.** Displays the category or categories by which the current Output view is filtered.

> ➤ **Viewed by.** Displays the name of the column on which you selected to drill down.

> ➤ **Export the view.** Saves the Output view to a specified file.

Refresh. Adds new log information that arrived in the Output window to the Filtered tab view.

Viewing the Vuser Script Log

During scenario execution, you can view a log containing run-time information about each running Vuser.

To view the Vuser script log for a particular Vuser:

1 In the Vusers dialog box, select the Vuser whose log you want to view, and click **Show Vuser Log**, or right-click the Vuser and select **Show Vuser Log**. The Vuser script log opens, displaying run-time information about the Vuser. This information is refreshed, by default, every 1000 milliseconds.
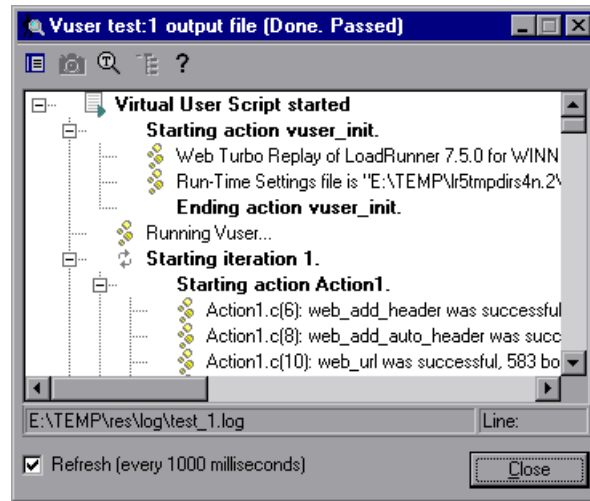
To change the default refresh settings, see "Options - Output Settings" on page 456.

2 Click **Close** to close the Vuser script log.

Understanding the Vuser Script Log

The Vuser Script Log displays run-time information about the Vuser. This information is refreshed, by default, every 1000 milliseconds.

Note: If you disabled the logging feature in the Run-Time Settings Log tab, the Vuser script log will contain output only if your script contains the **lr_output_message** or **lr_message** function. If you selected the **Send messages only when an error occurs** option in the Log tab, the Vuser script log will contain output only if there are script errors.

➤ **Show Text View.** Displays the run-time information in text format. To revert to the tree view, click the same button again.

➤ **Show Tree View.** Displays the run-time information in tree format. To revert to the text view, click the same button again.

➤ **Display.** Displays a snapshot of the Web page where an error occurred, when the error is highlighted in the Vuser log.

Note: To view a snapshot of the Web page where an error occurred, you must select the **Activate snapshot on error** option in the General tab of the Run-Time Settings dialog box before running the scenario.

➤ **Find Text.** Enter the text you want to search for in the Vuser log.

➤ **Expand Node.** Expands the node so that you can view additional run-time information details about the Vuser. To revert to the collapsed tree view, click this button again.

➤ **Collapse Node.** Collapses the node. To revert to the expanded tree view, click this button again.

➤ **Refresh (every 1000 milliseconds).** Instructs LoadRunner to refresh the run-time information displayed every 1000 milliseconds. Clear the **Refresh** check box to disable the log refreshing.

Note: You can change the default refresh settings in the "Options - Output Settings" on page 456.

➤ **Copy.** Enables you to copy text from the Vuser log. Right-click the selected text in the Vuser log, and select **Copy**.

➤ **Copy path from status bar.** Enables you to copy the path of the Vuser log. Right-click the path in the status bar, and select **Copy path from status bar**.

➤ **Vuser Log message icons.**

➤ **Start User Script.** Indicates the start of the virtual user script.

➤ **Start Iteration.** Indicates the start of an iteration.

➤ **End Iteration.** Indicates the end of an iteration.

➤ **Start/End Transaction.** Indicates the start or the end of a transaction.

➤ **Action.** Displays the name and description of an action.

➤ **Notifications.** Provides action information.

➤ **Errors.** Indicates that the Vuser encountered a problem, but test execution continued. Displays the error code and a description of the error.

Logging Execution Notes

The Controller provides you with a dialog box in which you can log comments while a scenario is running.
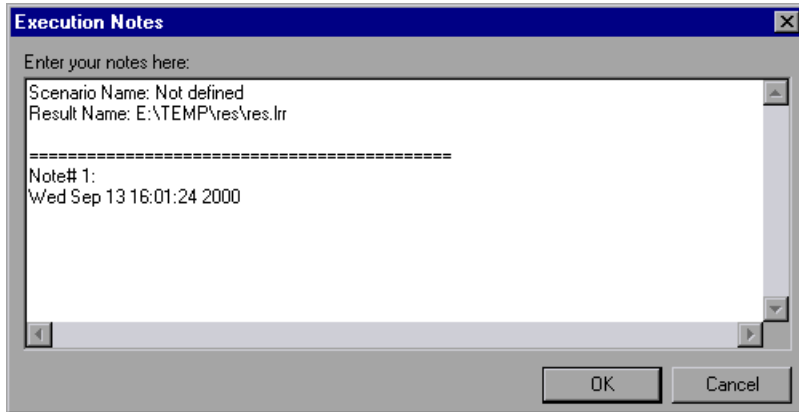
To log execution notes:

 1 Select **Scenario** > **Execution Notes**. The Execution Notes dialog box opens.

 2 Enter the notes that you want to log.

 3 Click **OK** to close the dialog box. LoadRunner saves the notes you recorded.

Viewing the Agent Summary

When you run a scenario with non-GUI Vusers, the machine running the Vusers invokes an agent that controls the Vuser execution on that load generator. During scenario execution, the agent displays a summary of the Ready, Running, and Paused Vusers.

The Agent window opens at the start of the scenario. You can minimize and restore it at any time.
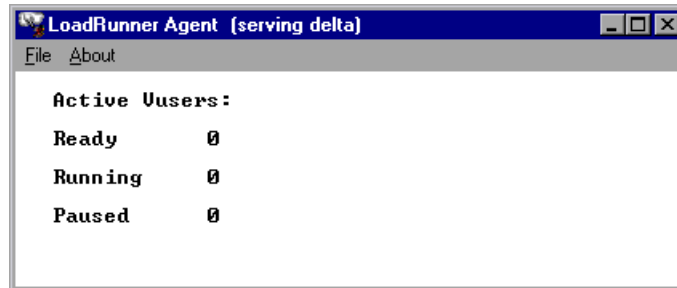
Part IV

Working with Firewalls

17

Working with Firewalls in LoadRunner

You can set up your LoadRunner system to run Vusers and monitor servers over a firewall.

This chapter includes:

About Using Firewalls in LoadRunner

Working with a firewall means that you can prevent unauthorized access to or from a private network, on specific port numbers.

For example, you can specify that there is no access to any port from the outside world, with the exception of the mail port (25), or you can specify that there is no outside connection from any ports to the outside except from the mail port and WEB port (80). The port settings are configured by the system administrator.

In a regular LoadRunner load test scenario (not over a firewall), the Controller has direct access to the LoadRunner agents running on remote machines. This enables the Controller to connect directly to those machines.
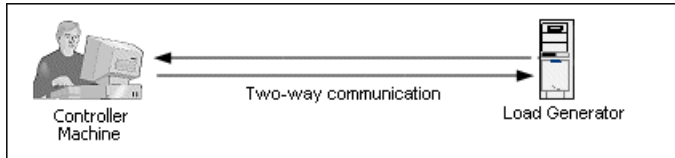
When running Vusers or monitoring applications over a firewall, this direct connection is blocked by the firewall. The connection cannot be established by the Controller, because it does not have permissions to open the firewall.
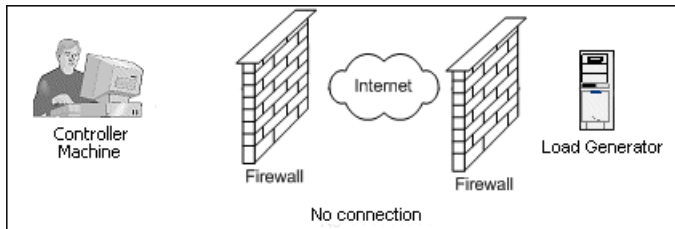
LoadRunner solves this problem by using a communication configuration based on HTTPS or secured TCP/IP. This configuration uses the standard SSL port on the firewall (port 443). For more information on HTTPS and TCP/IP system configuration, see "Setting up your Deployment (TCP or HTTPS)" on page 310.

A LoadRunner agent is installed on load generators running Vusers over a firewall, and on Monitor Over Firewall machines that monitor the servers that are located over a firewall. The agent communicates with the MI Listener machine through port 443 in the firewall.

The MI Listener is a component that serves as router between the Controller and the LoadRunner agent.
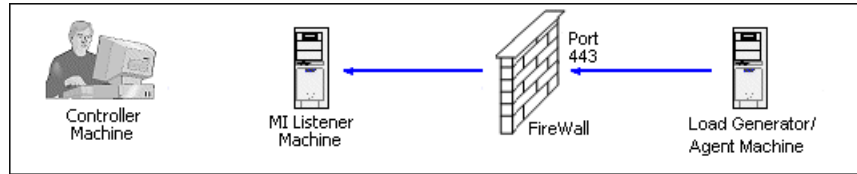
When the LoadRunner agent connects to the MI Listener, the MI Listener keeps a listing of the connection to the agent using a symbolic name that the agent passed to it.

When the Controller connects to the MI Listener, it communicates to the MI Listener through port 50500.
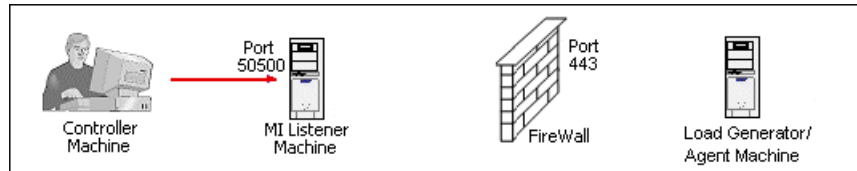

The Controller uses a symbolic name for the agent, and provides the MI Listener machine's name. If there has been a connection from the agent with the same symbolic name to this MI Listener, the connection is made between the Controller and the agent. Once you have a connection with the agent, you can run or monitor Vusers over a firewall.
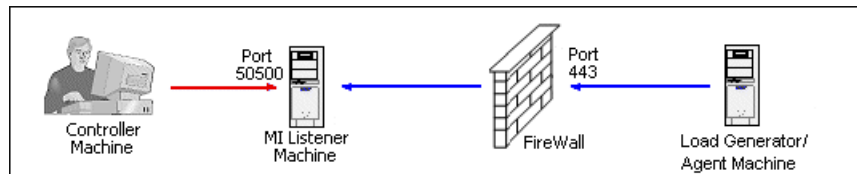
Example: Over Firewall Deployment

The following diagram is a basic example of a LoadRunner deployment over a firewall.
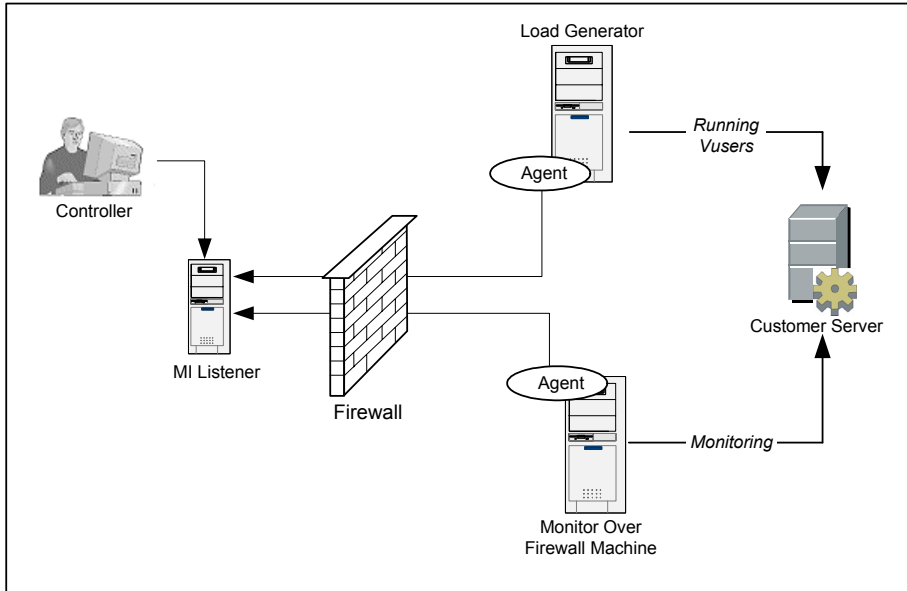
As explained in the previous section, the LoadRunner agent is installed on both the load generator machine and the Monitor Over Firewall machine. During installation, the LoadRunner agent is added as a Windows service.

The MI Listener serves as a router between:

➤ the agent on the load generator machine and the Controller, enabling the Controller to run Vusers over a firewall.

➤ the agent on the Monitor Over Firewall machine and the Controller, enabling the Controller to monitor the servers that are located over a firewall.

Note: You can connect the Controller to load generators over a firewall or Monitor Over Firewall machines without using the MI Listener and LoadRunner agent. To do so, open port 54345 on a firewall in the load generator/Monitor Over Firewall machine's LAN and in the Controller's LAN to allow incoming and outgoing data.

Setting Up your System to Use Firewalls: Basic Steps

Setting up your system to use firewalls involves the following stages of configuration:

➤ **Installation and initial configuration.** Installing the necessary components and doing initial configuration settings. See "Installing Over-Firewall Components" on page 308 and "Initial Configuration of the Over-Firewall System" on page 309.

➤ **Running Vusers over a firewall**. Setting up your system to run Vusers over a firewall when there is a firewall between the Controller and load generator host machines. See "Running Vusers Over a Firewall" on page 315.

➤ **Monitoring over a firewall.** Setting up your system to monitor the application under test (AUT) when there is a firewall between the Controller and AUT. See "Monitoring Over a Firewall" on page 316.

➤ **Checking Connectivity.** After installing and configuring all the necessary components, check that you are be able to establish a connection between the LoadRunner agent, MI Listener, and the Controller machine. See "Checking Connectivity" on page 326.

The following flow chart provides a general outline of the steps that you need to carry out to set up your system to use firewalls.
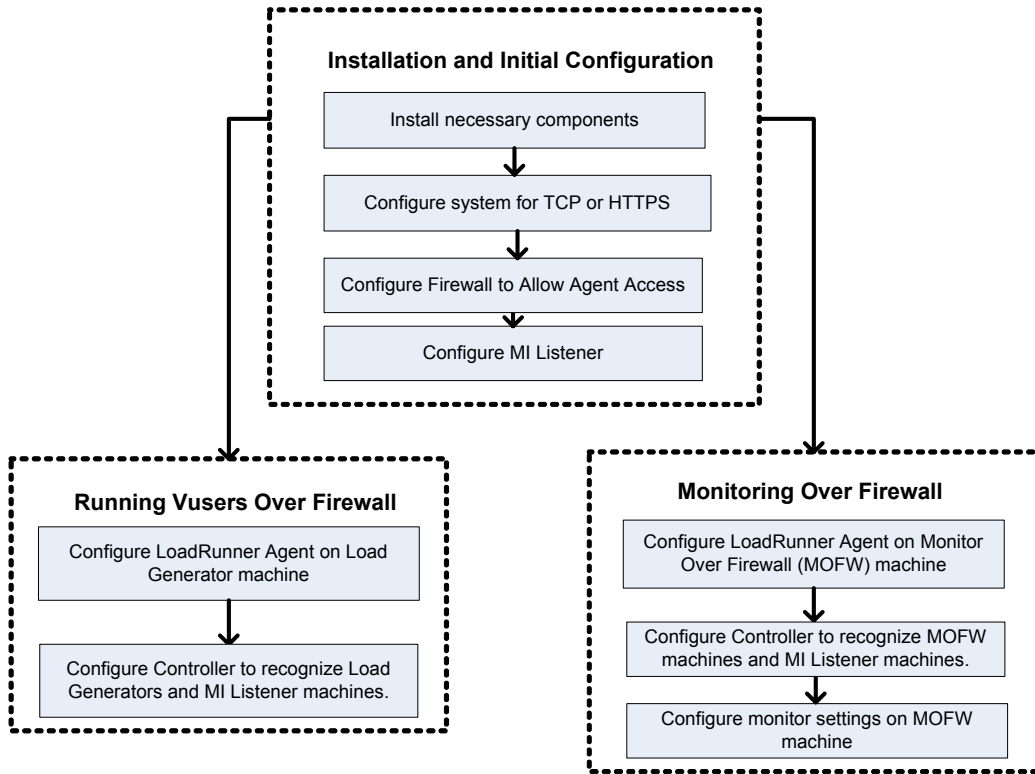
Installing Over-Firewall Components

To enable over-firewall communication, ensure that you have installed the following LoadRunner components:

➤ **MI Listener.** Serves as a router between the Controller and the LoadRunner agent. You install the MI Listener component on a dedicated machine. For installation instructions, refer to the *HP LoadRunner Installation Guide*.

For instructions on configuring the MI Listener machine, see "Configuring the MI Listener" on page 313.

> **Note:** You can also use the Controller machine as the MI Listener without the need for a separate installation. When acting as the MI Listener, the Controller machine cannot have Vusers running on it. In this case, the Controller must be a pure Controller and not a Controller + Load Generator.

➤ **Monitor Over Firewall component.** Used to monitor the servers that are located over a firewall. You install the Monitors over Firewall component on a dedicated machine. For installation instructions, refer to the *HP LoadRunner Installation Guide*.

For information about configuring the Monitor Over Firewall machine, see "Monitoring Over a Firewall" on page 316.

Initial Configuration of the Over-Firewall System

After you have installed the necessary components as described in "Installing Over-Firewall Components" on page 308. you are ready to configure your over-firewall system.

To perform initial configuration of your over-firewall system:

1 Configure your system according to TCP or HTTPS.

See "Setting up your Deployment (TCP or HTTPS)" on page 310.

2 Modify your firewall settings to enable communication between the machines on either side of the firewall.

See "Configuring the Firewall to Allow Agent Access" on page 312.

3 Configure the MI Listener.

See "Configuring the MI Listener" on page 313.

Setting up your Deployment (TCP or HTTPS)

To run Vusers or monitor servers over the firewall, configure your system according to one of the following configurations. Note that these configurations contain a firewall on each LAN. There may also be configurations where there is a firewall for the Over-Firewall LAN only.
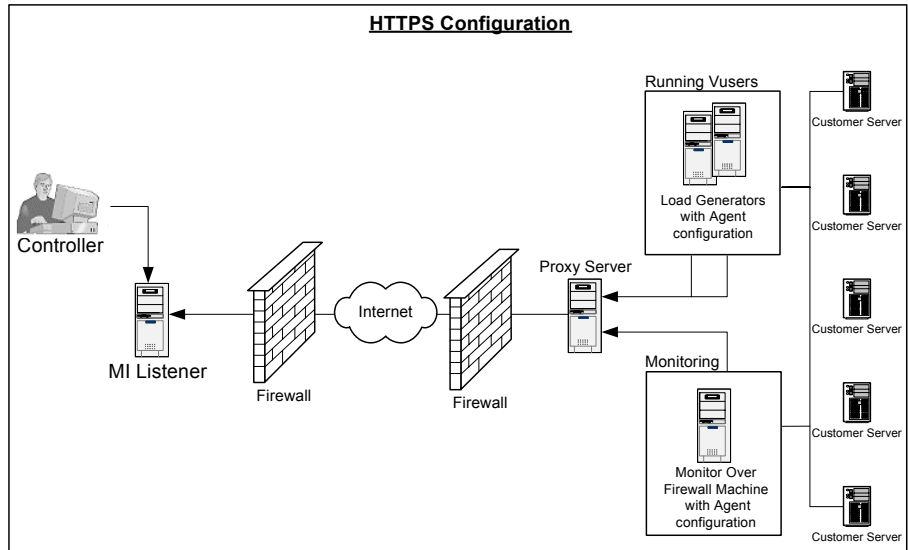
➤ **TCP Configuration**

The TCP configuration requires every LoadRunner agent machine behind the customer's firewall to be allowed to open a port in the firewall for outgoing communication.
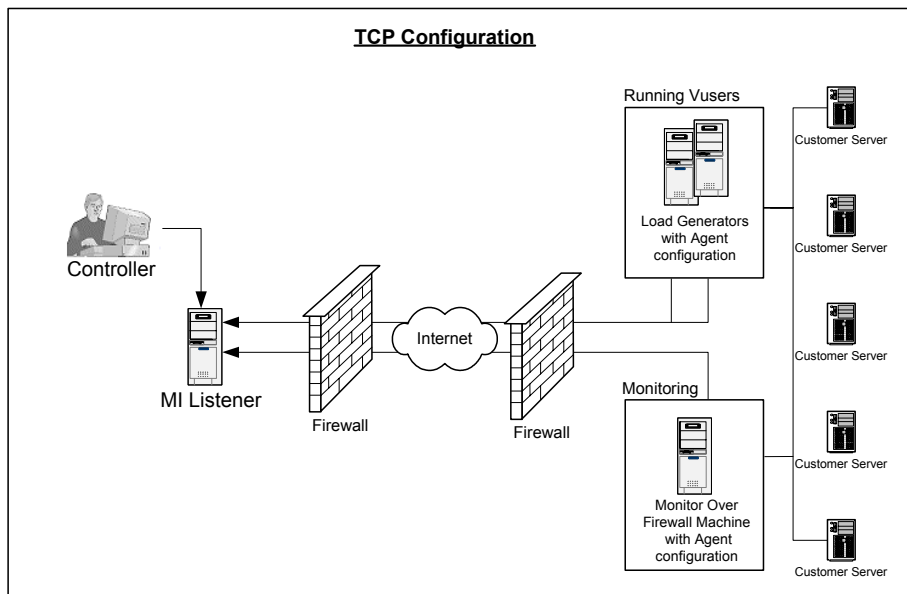
➤ **HTTPS Configuration**

In the HTTPS configuration, only one machine (the proxy server) is allowed to open a port in the firewall. Therefore it is necessary to tunnel all outgoing communications through the proxy server.

Configuring the Firewall to Allow Agent Access

You modify your firewall settings to enable communication between the machines inside the firewall and machines outside the firewall.

TCP Configuration

The LoadRunner agent attempts to establish a connection with the MI Listener using port 443, at intervals specified in the Connection Timeout field in the Agent Configuration dialog box. To enable this connection, allow an outgoing connection for HTTPS service on the firewall for port 443. The agent can then connect to the MI Listener, and the MI Listener can connect back to the agent. From this point on, the agent listens to commands from the MI Listener.

HTTPS Configuration

The LoadRunner agent attempts to establish a connection with the MI Listener, using the proxy port specified in the Proxy Port field, and at intervals specified in the Connection Timeout field in the Agent Configuration dialog box. When the connection is established, the proxy server connects to the MI Listener. To enable this connection, allow an outgoing connection for HTTPS service on the firewall for port 443. The proxy server can then connect to the MI Listener, and the MI Listener can connect back to the agent through the proxy server. From this point on, the agent listens to commands from the MI Listener.

Configuring the MI Listener

To enable running Vusers or monitoring over a firewall, you need to install the MI Listener on one or more machines in the same LAN as the Controller outside the firewall. For installation instructions, refer to the *HP LoadRunner Installation Guide*.

Notes:

➤ The Controller installation automatically includes the MI Listener, so you can designate the Controller as the MI Listener machine.

➤ The MI Listener can only be installed on Windows machines.

To configure the MI Listener:

1 Open incoming HTTPS service for port 443. The port settings are set by your system administrator.

2 Stop the LoadRunner agent on the MI Listener machine by right-clicking its icon in the system tray and selecting **Close** from the popup menu.

3 Run **MI Listener Configuration** from
Start > **Programs** > **LoadRunner** > **Advanced Settings**, or run <**LoadRunner root folder**>**\launch_service\bin\MILsnConfig.exe**.

313
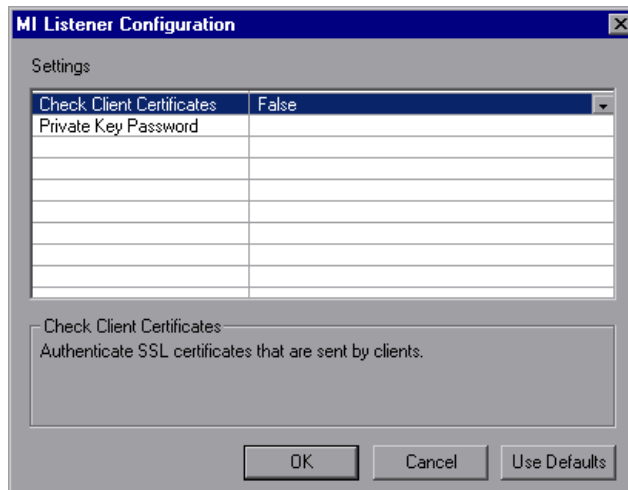
4 Set each option as described in the following table:

Option	Default Value	Description
Check Client Certificates	False	Choose True to request that the client send an SSL certificate when connecting, and to authenticate the certificate.
Private Key Password	None	The password that may be required during the SSL certificate authentication process.

5 Click **OK** to save your changes, **Cancel** to cancel them, or **Use Defaults**.

6 Restart the LoadRunner agent by double-clicking the shortcut on the desktop, or choosing **Start** > **Programs** > **LoadRunner**.

7 Make sure that port 443 is free on the MI Listener machine.

Note: Ensure that no Web Servers are running on the MI Listener or Monitor over Firewall machine. These servers use port 443 and will not allow the access required by the listening and monitoring processes.

Running Vusers Over a Firewall

Important: Before you configure your system to run Vusers over the firewall, ensure that you have completed the configuration steps described in "Initial Configuration of the Over-Firewall System" on page 309.

To set up your system to run Vusers over a firewall:

1 On each load generator machine that will be running over a firewall, configure the LoadRunner agent to communicate with the MI Listener.

For information on how to configure the LoadRunner agent, see "Configuring LoadRunner Agents Over the Firewall" on page 317.

2 Configure the Controller machine to recognize the load generator and MI Listener machines.

To run Vusers over a firewall, you need to create a unique connection between the Controller and the load generator machines that will be running over a firewall.

See "Configuring the Controller for Running over a Firewall" on page 323.

Monitoring Over a Firewall

Important: Before you configure your system to monitor your servers over a firewall, ensure that you have completed the configuration steps described in "Initial Configuration of the Over-Firewall System" on page 309.

To set up your system to monitor servers over a firewall:

1 On each Monitor Over Firewall machine, configure the LoadRunner agent to communicate with the MI Listener.

For information on how to configure the LoadRunner agent, see "Configuring LoadRunner Agents Over the Firewall" on page 317.

2 Configure the Controller machine to recognize the Monitor Over Firewall machines and MI Listener machines.

To monitor Vusers over a firewall, you need to create a unique connection between the Controller and the Monitor Over Firewall machines.

See "Configuring the Controller for Running over a Firewall" on page 323.

3 Use the Server Monitor configuration tool to configure the servers to monitor and to define specific measurements that LoadRunner collects for each monitored server.

You configure the monitor settings from the Monitor Over Firewall machine, using the Monitor Configuration tool
(**Start** > **Programs** > **LoadRunner** > **Advanced Settings** >
Monitor Configuration).

You select the type of monitors to run and the server whose resources you want to monitor, add the measurements to monitor for each server, and specify the frequency with which you want the monitored measurements to be reported.

For detailed information about how to use the Monitor Configuration tool, see Chapter 18, "Configuring Monitors Over a Firewall."

Configuring LoadRunner Agents Over the Firewall

On each load generator machine that will be running over a firewall and on each Monitor Over Firewall machine, you configure the LoadRunner agent to communicate with the MI Listener. The MI Listener serves as a router between the LoadRunner agent and the Controller.

This section includes the following topics:

➤ "Configuring the Windows LoadRunner Agent" on page 317

➤ "Configuring and Running the UNIX LoadRunner Agent" on page 319

➤ "Agent Configuration Settings" on page 321

Configuring the Windows LoadRunner Agent

This section includes instructions on how to configure the LoadRunner agent on Windows machines to communicate with the MI Listener.

To configure the LoadRunner agent on Windows machines:

1 Stop the LoadRunner agent by right-clicking its icon in the system tray and selecting **Close**.

2 Run **Agent Configuration** from **Start** > **Programs** > **LoadRunner** > **Advanced Settings**, or run **<LoadRunner root>\launch_service\bin\AgentConfig.exe**.

3 Select the **Enable Firewall Agent** check box, and then click **Settings**.

The Agent Configuration dialog box opens.

Agent Configuration		
Over Firewall Settings		

Property	Value
MI Listener Name	
Local Machine Key	
Connection Timeout (seconds)	20
MI Listener User Name	
MI Listener Password	xxxxx
Server Domain	
⊟ Connection Type	
⊙ TCP	
⊟ ○ HTTP	
Proxy Name	
Proxy Port	
Proxy User Name	
Proxy Password	xxxxx

MI Listener Name

The name, full name or IP address of the redirection server
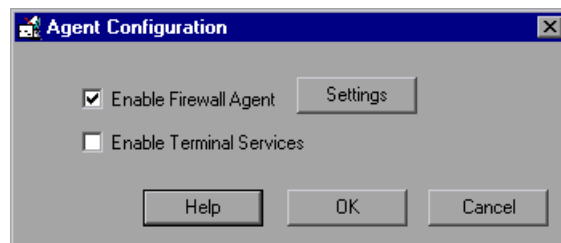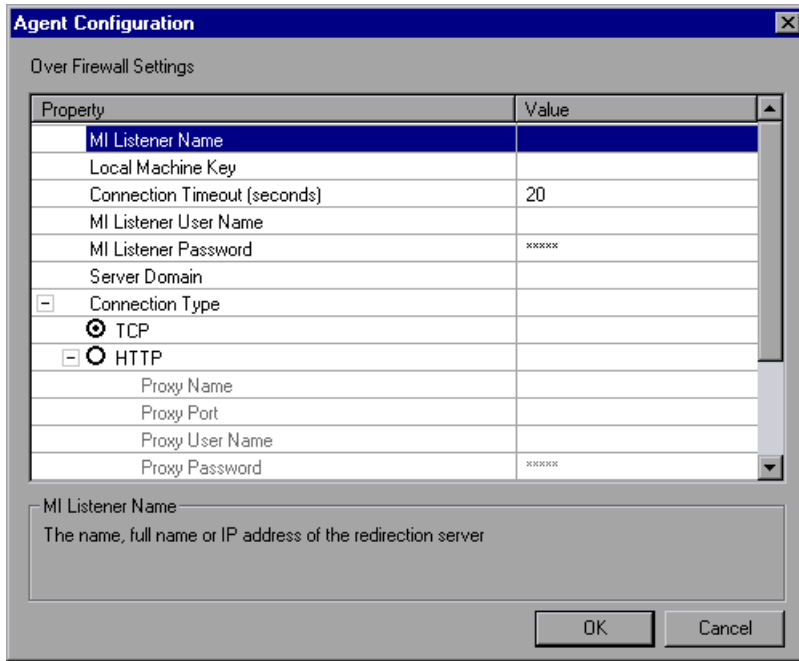
[OK]   [Cancel]

4 Set each option as described in "Agent Configuration Settings" on page 321.

5 Click **OK** to save your changes, or **Cancel** to cancel them.

6 Restart the LoadRunner agent by double-clicking the shortcut on the desktop, or select **Start** > **Programs** > **LoadRunner** > **LoadRunner Agent Service/Process**.

7 Check the connection status between the LoadRunner agent and the MI Listener.

A green light illuminated next to the LoadRunner Agent icon in the system tray indicates a successful connection between the LoadRunner agent and the MI Listener. A red light indicates that there is no connection between the agent and the MI Listener.

Configuring and Running the UNIX LoadRunner Agent

Load generator hosts can be installed on UNIX machines. This section describes how to configure and run LoadRunner agents on UNIX machines.

To configure the LoadRunner agent on UNIX machines:

1 Open **<LoadRunner root folder>/dat/br_lnch_server.cfg** in a text editor.

2 In the Firewall section, set FireWallServiceActive to 1 and save your changes.

3 Run **agent_config** from the **<LoadRunner root folder>/bin** directory to display the following menu:

```
Menu:
1. Show current settings.
2. Change a setting.
3. Save changes and exit.
4. Exit without saving.
5. Use default values.
```

4 Enter 1 to display the current settings:

```
Settings:
---------
1. MI Listener Name =
2. Local Machine Key =
3. Connection Timeout (seconds) = 20
4. Connection Type = TCP
5. Use Secure Connection (SSL) = False
6. Check Server Certificates = False
7. Client Certificate Owner = False
8. Private Key User Name =
9. Private Key Password =
10. Proxy Name =
11. Proxy Port =
12. Proxy User Name =
13. Proxy Password =
14. Proxy Domain =


Menu:
1. Show current settings.
2. Change a setting.
3. Save changes and exit.
4. Exit without saving.
5. Use default values.
```

5 To change a setting, enter 2 to display the settings menu:

```
Settings:
---------
1. MI Listener Name =
2. Local Machine Key =
3. Connection Timeout (seconds) = 20
4. Connection Type = TCP
5. Use Secure Connection (SSL) = False
6. Check Server Certificates = False
7. Client Certificate Owner = False
8. Private Key User Name =
9. Private Key Password =
10. Proxy Name =
11. Proxy Port =
12. Proxy User Name =
13. Proxy Password =
14. Proxy Domain =

Enter number of setting to change or 0 to go back to menu.
```

Enter the setting and continue according to the menu instructions. Set each option according to the "Agent Configuration Settings" on page 321.

6 Restart the agent for the configuration changes to take effect.

To restart the LoadRunner agent:

1 To remove the LoadRunner agent, run the command m_daemon_setup -remove from the **<LoadRunner root folder>/bin** directory.

Note: When the LoadRunner agent is configured to run over a firewall, and the agent is connected to the MI Listener, a file called <local_machine_key>_connected_to_MI_Listener is created in the temporary directory of the LoadRunner agent machine. The file is removed when the LoadRunner agent disconnects from the MI Listener.

---

2 To start the LoadRunner agent, run the command m_daemon_setup -install from the **<LoadRunner root folder>/bin** directory.

For more information about running the LoadRunner agent, see "UNIX Shell" on page 468.

Agent Configuration Settings

The following table provides an explanation of the agent configuration settings:

Option	Default Value	Description
MI Listener name	none	The name, full name, or IP address of the MI Listener.
Local Machine Key	none	A symbolic string identifier used to establish a unique connection between the Controller host behind the firewall, and the agent machine (via the MI Listener machine).
Connection Timeout (seconds)	20 seconds	The length of time you want the agent to wait before retrying to connect to the MI Listener machine. If zero, the connection is kept open from the time the agent is run.
MI Listener User Name	none	The username needed to connect to the MI Listener machine.
MI Listener Password	none	The password needed to connect to the MI Listener machine.
Server Domain	none	The domain name needed to connect to the MI Listener machine. This field is only required if NTLM is used.
Connection Type - TCP/HTTP	TCP	Select either **TCP** or **HTTP**, depending on the configuration you are using.
Connection Type - HTTP Proxy Name	none	The name of the proxy server. This option is mandatory if the **Connection Type** option is set to **HTTP.**

Option	Default Value	Description
Connection Type - HTTP Proxy Port	none	The proxy server connection port. This option is mandatory if the **Connection Type** option is set to **HTTP**.
Connection Type - HTTP Proxy User Name	none	The username of a user with connection rights to the proxy server.
Connection Type - HTTP Proxy Password	none	The user's password.
Connection Type - HTTP Proxy Domain	none	The user's domain if defined in the proxy server configuration. This option is required only if NTLM is used.
Use Secure Connection (SSL)	Disabled	Enable to connect using the Secure Sockets Layer protocol.
Use Secure Connection (SSL) - Check Server Certificates	None	Authenticates the SSL certificates that are sent by the server. Select **Medium** to verify that the server certificate is signed by a trusted Certification Authority. Select **High** to verify that the sender IP matches the certificate information. This setting is only available if **Use Secure Connection** is set to **True**.

Option	Default Value	Description
Use Secure Connection (SSL) - Private Key Password	none	The password that may be required during the SSL certificate authentication process. This option is relevant only if the **Client Certificate Owner** option is enabled.
Use Secure Connection (SSL) - Use Client Certificate	Disabled	Enable to load the SSL certificate (if required by the server to allow the connection to be made). This option is relevant only if the **Use Secure Connection** option is enabled.

Configuring the Controller for Running over a Firewall

To run or monitor Vusers over a firewall, you need to create a unique connection between the Controller and the agent machines. Agent machines include load generator machines that will be running over a firewall and all Monitor Over Firewall machines.

This connection is made through the MI Listener, which serves as router between the Controller and the LoadRunner agent. To establish this connection, you configure the Controller machine to define the agent machine as a load generator.

To configure the Controller for running vusers or monitoring over the firewall:

 1 Run the Controller from
   Start > **Programs** > **LoadRunner** > **Applications** > **Controller** and create a new scenario, or load an existing one.

 2 Click **Generators** to display the Load Generators window. In the Name field, enter the symbolic name of the server. This is the same name that you entered in the Local Machine Key setting in the Agent Configuration. (See "Agent Configuration Settings" on page 321).

In the example below, the server name is gumbi.

If the server is a UNIX server, change the Platform field to **UNIX**.
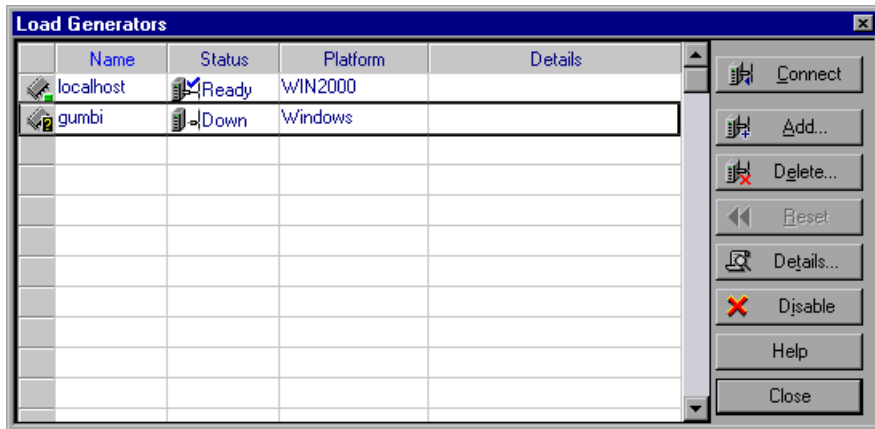
3 Select the Load Generator, and click **Details** to display the Load Generator Information.
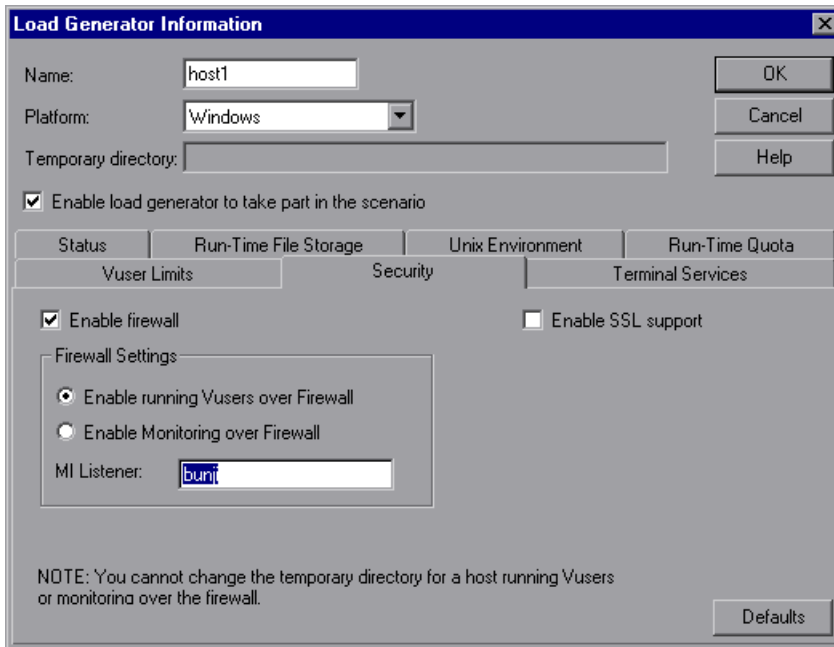
4 In the Security tab, enter the MI Listener machine's name in the MI Listener field. This is the same name that you entered in the MI Listener Name setting of the Agent Configuration dialog box. In this example, the MI Listener is bunji.

5 In the **Firewall Settings** section, select one of the following options:

➤ **Enable running Vusers over Firewall.** To run Vusers over the firewall.

➤ **Enable Monitoring over Firewall.** To monitor Vusers over the firewall.

6 Click **OK** to return to the Load Generators dialog box.

7 Select the load generator and click **Connect**.

Note: Remember that you cannot change the temporary directory on the host running or monitoring Vusers over the firewall.
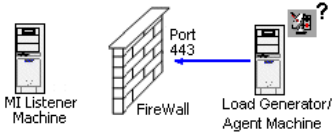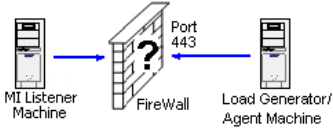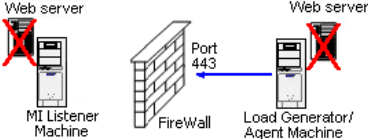
If you encounter connectivity problems, see "Checking Connectivity" on page 326. For other firewall troubleshooting, see "Troubleshooting Firewalls" on page 477.
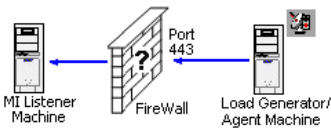
Checking Connectivity
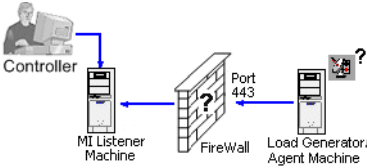
To run Vusers or monitor servers over a firewall, you must be able to establish a connection between the LoadRunner agent, MI Listener, and the Controller machine.

If you encounter connectivity problems after installing and configuring all the necessary components, check the table below for troubleshooting tips.

For additional firewall troubleshooting, see "Troubleshooting Firewalls" on page 477.

Check	Solution
To check that the Firewall service was activated on the agent machine: 	There should be a traffic light on the right side of the LoadRunner Agent icon on the machine running/ monitoring Vusers over a firewall. If there is no traffic light, this indicates that the 'FirewallServiceActive=1' is not set in the [FireWall] section of the Agent Settings. See "Configuring LoadRunner Agents Over the Firewall" on page 317.
To check that port 443 is open: 	On the agent machine, open a Command Prompt window, and type the following: telnet <MI_Listener_IP>443. For example: telnet 111.111.111.1111 443. If port 443 is open, a new Telnet window will open. If port 443 is not open, contact your network administrator.
To check that port 443 is available: 	If a Web server is running on the MI Listener or Monitor over Firewall machine, port 443 will not allow the access required by the listening and monitoring processes. Contact your network administrator to change the Web server port.

Check	Solution
To check connectivity between the agent and the MI Listener, when running the LoadRunner agent as a service: 	If there is a red light on the right side of the LoadRunner Agent icon when running the LoadRunner agent as a service, do the following: ➤ Check that port 443 is open. See the troubleshooting tip above. ➤ Check that the Agent Settings and Agent Configuration are correctly set. See "Configuring LoadRunner Agents Over the Firewall" on page 317. ➤ Run the agent as a Process. Launch < **Installation>\Launch_service\bin\magent proc.exe**. If this works, this indicates an authentication issue with the LoadRunner Agent Service. Browse to the **Service** > **LoadRunner Agent Service**, and change the properties of this service to System User Account or provide the username and password of someone who has administrative privileges on this machine.

Check	Solution
To check connectivity between the agent and the Controller, when monitoring over a firewall 	➤ Check that you entered the servers that you want to monitor in the Monitor Configuration dialog box. (See "Configuring Server Monitor Properties" on page 330). ➤ Start the LoadRunner Agent Process on the Monitor Over Firewall machine. (See "Configuring LoadRunner Agents Over the Firewall" on page 317). ➤ On the Controller, enter the name of the Monitor Over Firewall machine in the Load Generators dialog box, and click **Connect**. After about a minute, data should start streaming in from the Monitor Over Firewall machine through the MI Listener to the Controller. (See "Monitoring Over a Firewall" on page 316). ➤ If no data arrives at the Controller, try connecting the Controller to the MI Listener as if the Listener were used as a load generator. This will help identify the cause of the problem. Examine the log file on the Monitor Over Firewall machine by right-clicking the LoadRunner Agent icon. There should be no error messages. ➤ Start the MI Listener, and then manually start the LoadRunner Agent Process by running < **installation>\launch_service\bin\magnetproc.exe** on the Monitor Over Firewall machine. Allow the Monitor Over Firewall machine sufficient time to connect to the MI Listener, then connect the Controller to the Monitor Over Firewall machine. If the LoadRunner Agent Process crashes, either restart the agent or reboot the Monitor Over Firewall machine.

18

Configuring Monitors Over a Firewall

To monitor servers over a firewall, you need to configure the Monitor Over Firewall machine. On the Monitor Over Firewall machine, you use the Server Monitor configuration tool to select which servers to monitor and to define specific measurements that LoadRunner collects for each monitored server.

This chapter includes:

➤ About Monitoring over a Firewall on page 330
➤ Configuring Server Monitor Properties on page 330
➤ Configuring the Network Delay Monitor over a Firewall on page 337

About Monitoring over a Firewall

To enable monitoring of your servers over a firewall, you install the Monitor Over Firewall component on a dedicated machine.

Important: Before you configure your system to monitor your servers over a firewall, ensure that you have completed the configuration steps described in "Initial Configuration of the Over-Firewall System" on page 309.

After you have set up your LoadRunner system to work with firewalls, you need to configure the monitor settings on the Monitor Over Firewall machine.

You use the Server Monitor configuration tool to select which servers to monitor and to define specific measurements that LoadRunner collects for each monitored server. For more information, see "Configuring Server Monitor Properties" below.

Configuring Server Monitor Properties

After you have installed and configured the LoadRunner agent, the Monitors over Firewall component, the MI Listener, and the Controller machine, you need to choose the server measurements that you want the Monitor Over Firewall machine to monitor.

You configure the server monitor properties from the Monitor Over Firewall machine, using the Monitor Configuration dialog box. You can select the type of monitors to run and the server whose resources you want to monitor, add the measurements to monitor for each server, and specify the frequency with which you want the monitored measurements to be reported.

To configure server monitor properties:

1 Select **Start > Programs > LoadRunner > Advanced Settings > Monitor Configuration**. For machines without the complete LoadRunner installation, select **Start > Programs > Server Monitor > Monitor Configuration.** The Monitor Configuration dialog box opens.
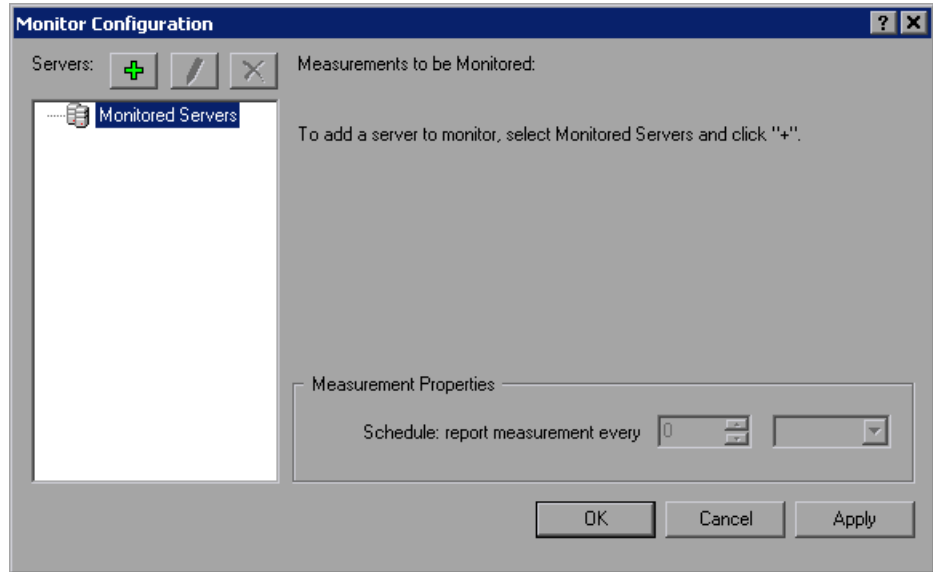
2 Click the **Add Server** button. The New Monitored Server Properties dialog box opens.
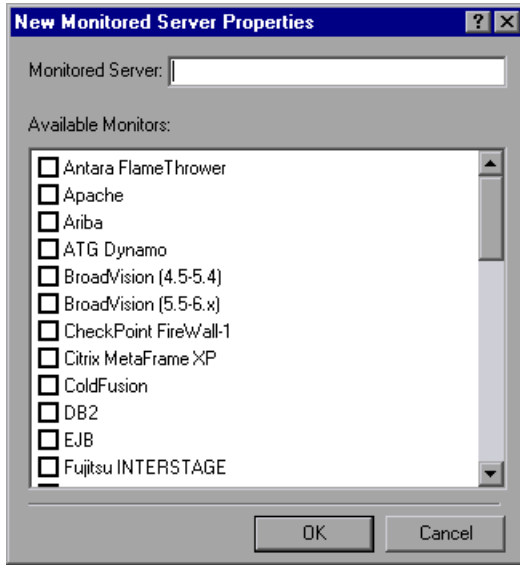
3 In the **Monitored Server** box, type the name or IP address of the server whose resources you want to monitor.

Note: To add several servers simultaneously, separate the server names or IP ranges with commas. For example: 255.255.255.0-255.255.255.5, server1, server2.

---

4 From the Available Monitors list, select the monitors appropriate for the server being monitored.

Note: Data can only be viewed for the monitors that are enabled with your LoadRunner license key. To preview your license key information, select **Start** > **Programs** > **LoadRunner**. HP LoadRunner opens. Click the **License** button to display the LoadRunner license information.

---

5 Click **OK** to close the New Monitored Server Properties dialog box. The Monitored Servers list is displayed in the Monitor Configuration dialog box.

For certain monitors, LoadRunner displays default measurements in the Measurements to be Monitored section. You can specify the frequency at which LoadRunner reports the measurement in the Measurement Properties section. For details on selecting measurements, see "Adding and Removing Measurements" on page 334.

6 To add additional monitored servers to the list, repeat steps 1-5.

7 To edit the monitor configuration properties for a server, click the **Edit** button. The Monitored Server Properties dialog box opens enabling you to edit the monitors for the server whose resources you are monitoring.

8 Click **Apply** to save your settings.

Adding and Removing Measurements

After you configure one or more server machines to monitor, you add measurements to monitor for each server. If LoadRunner added default measurements, you can edit them as required.

To add a measurement to monitor:

 1 Select a server from the Monitored Servers list.

 2 Click the **Add Measurement** button. Select the appropriate monitor. A dialog box opens, enabling you to select measurements for the monitor you selected.

 3 Select the measurements that you want to monitor, and click **OK**.

 4 Click **Apply** to save your settings.

For information on configuring measurements for each server monitor, see the relevant chapter in the *HP LoadRunner Monitor Reference*.

To remove a measurement from the measurements list:

 1 Select the measurement, and click the **Delete** button.

 2 Click **Apply** to save your settings.

Configuring Measurement Frequency

Once you have configured monitor measurements, you configure measurement frequency.

In the Measurement Properties section, you set a measurement schedule for each measurement to be reported.
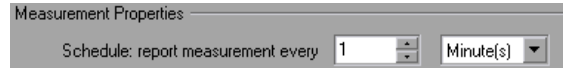
To set a measurement schedule for a measurement:

1 Select the configured server measurement you want to schedule.

2 Specify the frequency at which you want LoadRunner to report the measurement.

3 Click **Apply** to save your settings.

Cloning a Monitored Server's Properties

If you want to monitor the same properties on different server machines, you can clone a selected server's properties using the Clone Monitored Server Properties dialog box.

To clone a monitored server's properties:

1 In the Monitor Configuration dialog box, right-click the server you want to clone, and select **Clone**. The Clone Monitored Server Properties dialog box opens.
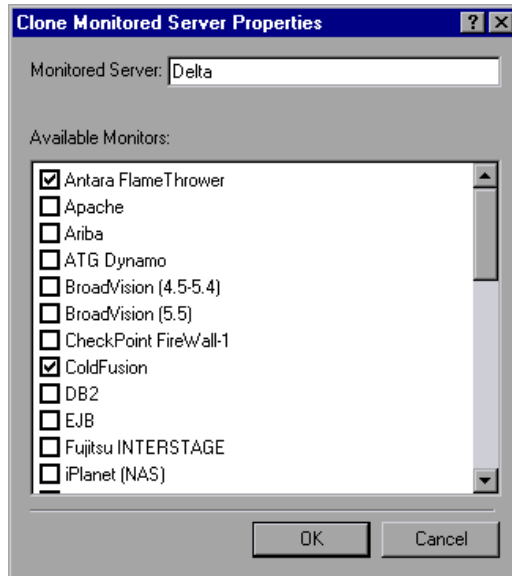
2 In the Monitored Server box, type the name or IP address of the clone server you want to create.

Note: To create several servers simultaneously, separate the server names or IP ranges with commas. For example: 255.255.255.0-255.255.255.5, server1, server2.

---

3 The **Available Monitors** list displays the monitors that were selected for the server being cloned. Select additional appropriate monitors for the clone server.

4 Click **OK** to close the Clone Monitored Server Properties dialog box. The cloned server is displayed in the **Monitored Servers** list.

5 Click **Apply** to save your settings.

Configuring the Network Delay Monitor over a Firewall

To run the Network Delay Monitor when there are firewalls between the Controller machine and the source machine, you must configure the Network Delay Monitor (see "Configuring the Network Delay Time Monitor" in the *HP LoadRunner Monitor Reference*), and add the following to step 3:

In the **Monitor the network delay from machine** section, enter the server name or IP address of the source machine according to the following format:

<MI Listener machine>:<source machine local key>.

where source machine local key is the Local Machine Key that you chose when configuring the LoadRunner agent on the source machine. (See "Agent Configuration Settings" on page 321.)

For example: 12.12.12.3:vds

Part V

Working with Diagnostics

19

LoadRunner Diagnostics Modules

LoadRunner's diagnostics modules provide detailed performance information to help you rapidly identify and pinpoint performance problems in Siebel, Oracle, SAP, J2EE, and .NET environments.

The diagnostics modules come with a set of customized Analysis reports. Using these reports, you can communicate your insights to developers and managers to accelerate resolution, while avoiding expensive hardware upgrades.

Note: This section describes working with ERP/CRM diagnostics modules.

For information on working with Diagnostics for J2EE & .NET, see the *HP Diagnostics User's Guide*.

This chapter includes:

About LoadRunner Diagnostics Modules

During a performance test, LoadRunner's diagnostics modules trace, time, and troubleshoot individual transactions across the Web, application, and database servers. You can drill-down from a slow end-user transaction all the way to the bottle necked method or SQL statement. The LoadRunner diagnostics modules enable organizations to:

➤ Trace the application components exercised by business processes.

➤ Rapidly isolate application components that have a significant impact on end-user experience.

➤ Provide developers with precise data on how to make performance improvements.

Such precision pinpointing of performance problems directly translates into significant business value:

➤ Quicker and more efficient performance testing cycles

➤ Decreased time to problem resolution by development

➤ Better performing applications that are optimized to meet the needs of the business

LoadRunner ERP/CRM Diagnostics Types

LoadRunner provides the following ERP/CRM Diagnostics solutions:

➤ **Siebel Diagnostics.** The diagram below shows transaction breakdown on a Siebel system.
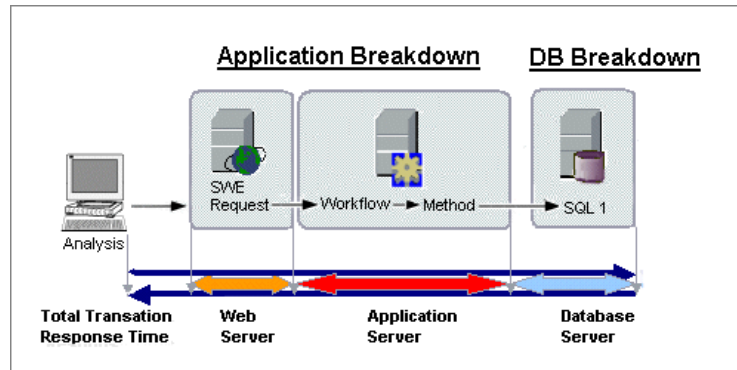
LoadRunner's Siebel Diagnostics are split into the following modules:

➤ **Siebel Diagnostics Module.** Enables you to break down Siebel transactions into layers, areas, sub-areas, servers, and scripts. You can also view the transaction chain of calls and call stack statistics to track the percent of time spent for each part of the transaction. Siebel-Web Vusers support Siebel Diagnostics. For more information, see Chapter 20, "Configuring Siebel Diagnostics."

➤ **Siebel DB Diagnostics Module.** Helps you to rapidly identify and resolve database performance problems. You can view the SQLs for each transaction, identify the problematic SQL queries of each script, and identify at what point problems occurred. Siebel-Web Vusers support Siebel DB Diagnostics. For more information, see Chapter 21, "Configuring Siebel DB Diagnostics."

➤ **Oracle 11i Diagnostics.** Oracle 11i Diagnostics helps pinpoint performance problems on Oracle NCA systems. The diagnostics information drills down from the transaction, to the SQL statements and the SQL stages of each statement. Oracle NCA Vusers support Oracle 11i Diagnostics. For more information, see Chapter 22, "Configuring Oracle 11i Diagnostics."

343

➤ **SAP Diagnostics.** SAP Diagnostics enables you to pinpoint the root cause of a certain problem (i.e. DBA, Network, WAS, Application, OS/HW) quickly and easily, and engage with the relevant expert only, without having to present the problem to a whole team of people. For more information, see Chapter 23, "Configuring SAP Diagnostics."

ERP/CRM Diagnostics Module Architecture

The ERP/CRM Diagnostics architecture is composed of the following components:
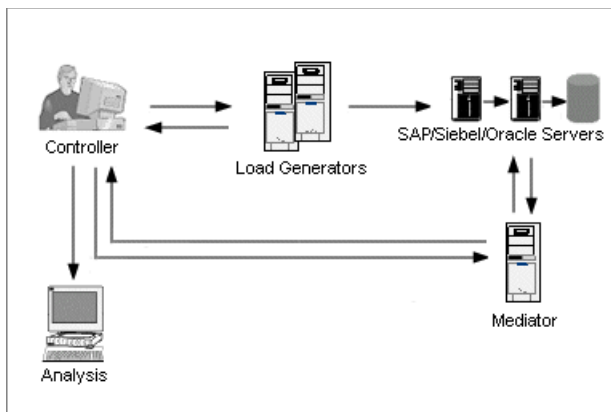
➤ **ERP/CRM Mediator.** The ERP/CRM Mediator ("Mediator") gathers and correlates offline transaction data from the Web, database, and application servers. The Mediator must be installed on a machine that resides in the same LAN as the monitored ERP/CRM server, and preferably on a dedicated server. We do not recommend installing the Mediator on a Siebel, Oracle, or SAP server that is involved in the load test. For more information on installing the Mediator, see the *HP LoadRunner Installation Guide*.

By default, the Mediator agent is installed to run as a process. We recommend configuring the Mediator agent to run as a service.

If you do run the agent as a process, you may encounter the following Microsoft Windows networking limitation: **Failed to establish connection. System error 1219**. You can do one of the following:

➤ Run the agent as a service. For more information, see "Running the LoadRunner Agent as a Service" on page 485.

➤ Disconnect all previous connections to the server and try again.

Note: For Siebel DB Diagnostics, it may take a long time to copy the files from the application server to the Mediator, and then from the Mediator to the results directory. During the first copying stage, the Summary Data Processing window is displayed. To minimize copying time in the second stage (if you are not working over a firewall), we recommend using localhost as the Mediator machine. This reduces the time it takes to copy the Siebel DB Diagnostics files to the results directory, since the diagnostics files are already on the Controller machine.

➤ **Controller.** Before scenario execution, the Controller transfers all server information to the Mediators and distributes the percentage of users that will participate in the monitoring. After scenario execution, the Controller collects the aggregated transaction data files from the Mediators and collates the results. Siebel results are transferred to the **\sbl_bd** directory, and Oracle 11i Diagnostics are transferred to the **\ora_bd** directory. SAP results are transferred to the **\sap_bd** directory. You can view the status of diagnostics file collation in the Collate Results dialog box. For more information, see "Collating Results" on page 231.

➤ **Load Generator.** When you execute a scenario, the Controller distributes each Vuser to a load generator, and the load generator executes the Vuser script.

➤ **Analysis.** Displays detailed diagnostics graphs and reports. For more information about the diagnostics graphs, see the *HP LoadRunner Analysis User's Guide*.

Working with LoadRunner ERP/CRM Diagnostics

To use LoadRunner's ERP/CRM Diagnostics, follow these steps:

 1 Prepare for generating diagnostics data.

Make sure that the ERP/CRM Mediator is installed. The Mediator collects and processes the diagnostics data.

The ERP/CRM Mediator is installed on the Controller machine as part of the LoadRunner Full Setup. For information on installing the ERP/CRM Mediator on a dedicated machine, see the *HP LoadRunner Installation Guide*.

 2 Configure the server machine to enable the diagnostics feature.

For more information, see "Configuring Siebel Diagnostics on the Application and Web Servers" on page 356, "Enabling Server Logging on the Siebel Server" on page 367, "Enabling Server Logging On the Oracle Server" on page 373, and "Selecting the Oracle NCA Application Version" on page 375.

 3 Prepare the Controller machine to generate diagnostics data and communicate with the Mediator machines.

For more information, see:

➤ Chapter 20, "Configuring Siebel Diagnostics"

➤ Chapter 21, "Configuring Siebel DB Diagnostics"

➤ Chapter 22, "Configuring Oracle 11i Diagnostics"

➤ Chapter 23, "Configuring SAP Diagnostics"

➤ "Enabling LoadRunner Diagnostics" on page 351

 4 Collect and prepare the diagnostics data.

During the load test, the Mediator collects the data and processes the diagnostics information.

 5 Create the results.

After the load test, the Controller collects the aggregated data from the Mediator machines and collates the results. For more information on result collation, see "Collating Results" on page 231.

6 Present the data.

Use the Analysis graphs to view the diagnostics data and drill down to the problematic areas. For more information about ERP/CRM diagnostics graphs, see the *HP LoadRunner Analysis User's Guide*.

Connecting to a Remote Server

➤ Connecting to a Remote Windows Server

➤ Connecting to a Remote UNIX Server

Connecting to a Remote Windows Server

For Windows platforms, the user should have administrator privileges.

When working on a remote WINDOWS server, the Mediator first tries to add a connection to that machine with the details supplied by the user. This configuration should give administrator permissions to the remote machine.

If the Mediator machine is already connected to the server machine with a different configuration, it will use the existing connection to the server. This may lead to an error if the user is a non-administrator. (See the troubleshooting chapter - Error 1219.)

To solve this conflict you may select one of the following solutions:

➤ Change the LoadRunner Agent to work as a service and not as a process.

For more information, see "Running the LoadRunner Agent as a Service" on page 485.

➤ Close existing connections from the Mediator to the server.

These connections can use **\\servername\sharename** in the Windows Explorer, or a specified mount in My Computer.

To check connections exist, open the Command prompt and run the following from command line:

net use

To remove a connection, remove the mount, and, from the Command line, run the following command:

net use \\servername\sharename /DELETE

➤ You may use your own created connection to the log directory by supplying the UNC-path to the log directory and marking the OS as WINDOWS. When the Mediator runs, it will not try to create a connection, but rely on the given UNC-path instead.

Connecting to a Remote UNIX Server

When working with a UNIX platform, the Mediator supports two types of connections:

➤ Remote Shell (RSH/RCP)

➤ Secured Shell (PLINK/PSCP)

Remote Shell (RSH/RCP)

Before using the remote shell (RSH) through the Mediator, please verify the following:

➤ RSH and RCP daemons should be running on the UNIX server.

➤ The user should have permission to run remote shell commands. To check this, type the following at the DOS command prompt:

rsh <server machine name> -l <UNIX user login name> -n <command>

For example: rsh my_unix -l my_name -n "cd ~;pwd"

Note: Only RSH commands that work from the DOS command prompt window should work with LoadRunner.

If you encounter an "RSH command failed" error in LoadRunner, try to run the same command from the DOS command prompt to verify if it is a problem with the command or with LoadRunner. If it runs in the DOS command prompt then the command is valid, and you should contact the HP Software Support Web site (www.hp.com/go/hpsoftwaresupport) for further help.

If you cannot run this command from the DOS command prompt either, then contact your UNIX administrator. For more information about RSH commands, see "UNIX Shell" on page 468.

➤ No output should be generated after executing the RSH command. Due to a bug in the RCP UNIX command, you should not generate output from the **.login**, **.profile**, and **.cshrc** files (for example by echo, or in any other way, including commands that generate output indirectly, such as biff). Where an existing user generates output in the RSH step that cannot be deleted, you should create a new user that does not generate output, and who has permissions to run RSH and RCP commands on the server machine.

Secured Shell (PLINK/PSCP)

The secured shell is based on the SSH protocol which enables a secure way of connecting to a remote machine by an authentication mechanism (using RSA / DSA key pair) and encrypted communication.

The SSH protocol has a few levels of security:

➤ Using a username and password.

➤ Using a username and key pair without passphrase protection.

➤ Using a username, key pair and a passphrase protected private key.

The Mediator uses the 'PuTTY' suite which is free implementation of SSH.

You can find the following PuTTY tools in the bin directory of the installation:

PUTTY.EXE, PAGEANT.EXE, PLINK.EXE, PSCP.EXE, PSFTP.EXE, PUTTYGEN.EXE and PUTTY.HLP

Using the Import command from the Conversions menu of PuTTYgen, you can load private key in OpenSSH's format and ssh.com's format and save it back out as a PuTTY-format key. For more information, please see the *PuTTY User Manual* which resides in the bin directory of the installation.

Before using the secured shell through the Mediator, please verify the following:

➤ SSH daemon should be running on the UNIX server.

➤ No output should be generated after executing the PuTTY commands. You should not generate output from the **.login**, **.profile**, and **.cshrc** files, (for example by echo, or by any other form including commands that generate output indirectly, such as biff). Where an existing user generates output that cannot be deleted, you should create a new user that does not generate output.

To verify that the PuTTY commands are working without generating any errors, type the following at the DOS command prompt under the bin directory of the installation:

PLINK.EXE <server machine name> -ssh -l <UNIX user login name> -i <private key full name if used> -pw <password or passphrase> <command>

For example:

Security Level A: PLINK.EXE my_unix -ssh -l my_name -pw "my_password" "ls"

Security Level B: PLINK.EXE my_unix -ssh -l my_name -i "my_private_key" -pw "" "ls"

Security Level C: PLINK.EXE my_unix -ssh -l my_name -i "my_private_key" -pw "my_passphrase" "ls"

Similarly check the **PSCP.EXE** command:

PSCP.EXE -scp -r -q <private key full name if used> -pw <password or passphrase> <local file name> <UNIX user login name>@<server machine name>:<remote file name>

If you encounter an "SSH command failed" error in LoadRunner, try to run the same command from the DOS command prompt to verify if it is a problem with the command or with LoadRunner. If it runs in the DOS command prompt then the command is valid, and you should contact the HP Software Support Web site (www.hp.com/go/hpsoftwaresupport) for further help.

If you cannot run this command from the DOS command prompt either, then contact your UNIX administrator.

Enabling LoadRunner Diagnostics

After setting up the diagnostics module and successfully testing the connection between the Controller and the Mediators, you enable the diagnostics module and specify the sampling percentage of transaction data to include in the diagnostics graphs.

Note: The Diagnostics Distribution dialog box is disabled during scenario execution. You must enable and configure the Diagnostics modules before running the scenario.

To enable collection of diagnostics data:

1 From the Controller menu, select **Diagnostics** > **Configuration**. The Diagnostics Distribution dialog box opens.
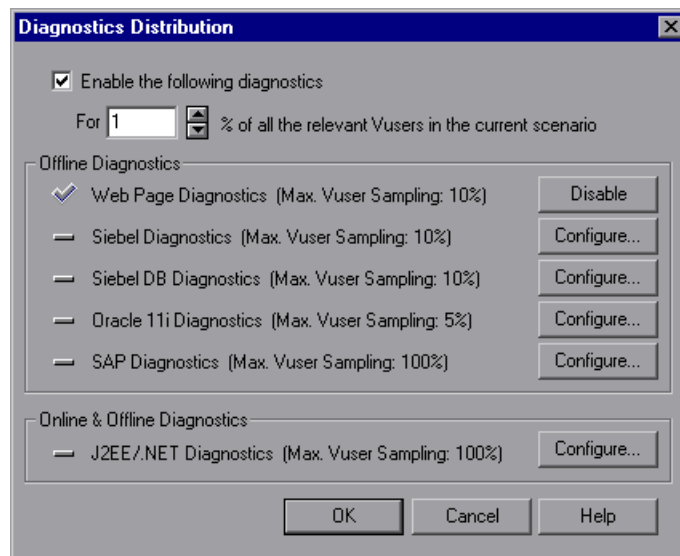
2 To enable diagnostics monitoring, select **Enable the following diagnostics** and set the percent of Vusers to participate in the monitoring as described in "Understanding the Diagnostics Distribution Dialog Box" on page 352.

3 By default, Web Page Diagnostics is enabled.

To enable and configure other offline diagnostics, click the **Configure** button next to the appropriate diagnostics type for which you want to collect diagnotcis data.

➤ To configure Siebel Diagnostics, see Chapter 20, "Configuring Siebel Diagnostics."

➤ To configure Siebel DB Diagnostics, see Chapter 21, "Configuring Siebel DB Diagnostics."

➤ To configure Oracle 11i Diagnostics, see Chapter 22, "Configuring Oracle 11i Diagnostics."

➤ To configure SAP Diagnostics, see Chapter 23, "Configuring SAP Diagnostics."

4 To enable and configure online and offline diagnostics, click the **Configure** button next to **J2EE/.NET Diagnostics (Max. Vuser Sampling: 100%)**.

To configure J2EE/.NET Diagnostics, see the *HP Diagnostics User's Guide*.

Understanding the Diagnostics Distribution Dialog Box

➤ **Enable the following diagnostics.** Enables LoadRunner to generate offline Web Page, Siebel, Siebel DB, Oracle 11i, and SAP Diagnostics graphs. Enables LoadRunner to generate online and offline J2EE & .NET Diagnostics graphs.

➤ **For X % of all the relevant Vusers in the current scenario.** Specify the percentage of Vusers for which you want to collect diagnostics data. This value determines how many of the transactions on the application server are reported to the Controller. Reducing this percentage will reduce the overhead on the application server for Web Page, Oracle 11i, and J2EE & .NET Diagnostics.

For example, if you enter a sampling value of 25% and run 12 Vusers in *group1*, 8 Vusers in *group2*, and 1 Vuser in *group3*, diagnostics data will be collected for 3 Vusers in *group1*, 2 Vusers in *group2*, and 1 Vuser in *group3*.

Note: The minimum percentage of vuser sampling allowed is 1%, or 1 virtual user per group, whichever is more. The maximum percentage of vuser sampling allowed is the lowest of the **Max. Vuser Sampling** values of all the selected diagnostics types.

➤ **Offline Diagnostics.** Click the appropriate **Configure** button to enable and configure the offline diagnostics module as follows:

   ➤ **Web Page Diagnostics.** Generates Web Page Diagnostics graphs, which provide you with performance information for each transaction and sub-transaction defined in your script. The maximum percentage of Vusers for which diagnostics data can be collected is 10%.

   ➤ **Siebel Diagnostics.** Generates Siebel Diagnostics graphs that allow you to break down Siebel transactions into layers, areas, sub-areas, servers, and scripts. This enables you to pinpoint the exact location where time is consumed. You can also view the transaction chain of calls and call stack statistics to track the percent of time spent for each part of the transaction. The maximum percentage of Vusers for which diagnostics data can be collected is 10%, or not more than 100 Vusers.

   ➤ **Siebel DB Diagnostics.** Generates Siebel DB Diagnostics graphs that allow you to break down transactions into their SQL statements and stages. This enables you to pinpoint problem areas in any layer of the Siebel database calls. The maximum percentage of Vusers for which diagnostics data can be collected is 10%.

   ➤ **Oracle 11i Diagnostics.** Generates Oracle 11i Diagnostics graphs that allow you to break down transactions into their SQL statements and stages. This enables you to pinpoint problem areas in any layer of the Oracle database calls. The maximum percentage of Vusers for which diagnostics data can be collected is 5%.

   ➤ **SAP Diagnostics.** Generates SAP Diagnostics graphs that allow you to break down transactions into DB, application processing, systems and interface performance levels. This enables you to pinpoint problem areas in any component of the SAP environment. The maximum percentage of Vusers for which diagnostics data can be collected is 100%.

➤ **Online and Offline Diagnostics.** Click the **Configure** button to enable and configure the online and offline diagnostics module as follows:

➤ **J2EE/.NET Diagnostics.** Provides online and offline J2EE & .NET Diagnostic graphs. In the online diagnostics, you can view the entire chain of activity on the server side of the system. You can break down J2EE/.NET layers into components and methods to enable you to pinpoint the exact location where time is consumed. You can also view the transaction chain of calls and call stack statistics to track the percent of time spent for each part of the transaction. The maximum percentage of Vusers for which diagnostics data can be collected is 100%.

After a scenario run, you can use the J2EE & .NET Diagnostics graphs to analyze server performance and generate reports. For more information, see the *HP Diagnostics User's Guide*.

Viewing Diagnostics Results

To view diagnostics results, in the Run tab of the Controller select **Results > Analyze Results**, or click the **Analyze Results** button. Analysis opens.

You can use the Analysis diagnostics graphs and reports to view the performance data and drill down to pinpoint problem areas in any layer of the application.

For more information about diagnostics graphs, see the *HP LoadRunner Analysis User's Guide*.

20

Configuring Siebel Diagnostics

Siebel Diagnostics enable you to break down Siebel transactions into layers, areas, sub-areas, servers, and scripts. You can also view the transaction chain of calls and call stack statistics to track the percent of time spent for each part of the transaction. Siebel-Web Vusers support Siebel Diagnostics.

This chapter includes:

➤ Configuring Siebel Diagnostics on the Application and Web Servers on page 356

➤ Configuring Diagnostics where the Web Server is Inside a DMZ on page 358

➤ Copying Files from the Siebel Application Server to the Mediator on page 359

➤ Setting Up the Siebel Diagnostics Module on page 360

Note: Siebel Diagnostics (Siebel Application Response Measurements) supports Siebel application servers versions 7.53 and 7.7.

Configuring Siebel Diagnostics on the Application and Web Servers

To configure Siebel Application and Web servers for diagnostics data collection, you perform the following:

➤ enable Siebel Diagnostics on all Siebel Application and Web servers involved in the load test

➤ optimize server performance settings

➤ generate a list of Siebel server IDs (required for Siebel Application servers only)

To enable Siebel Diagnostics:

1 Set the environment variable on the Siebel server to:

SIEBEL_SarmEnabled=true

2 Restart the server.

To optimize server performance:

You can change the maximum memory caching and file size using the following variables:

SIEBEL_SarmMaxMemory= <bytes>

SIEBEL_SarmMaxFileSize = <bytes>

SIEBEL_SarmMaxMemory controls the size of the buffer that Siebel keeps in the memory before writing the information to the Siebel log files. You can improve server performance by increasing the parameter value. However, information from the end of the run will be missing from the Analysis graphs. We recommend setting SIEBEL_SarmMaxMemory= 50000 for low loads on the server and SIEBEL_SarmMaxMemory= 1000000 for high loads on the server. A low load on the server is 20 Vusers or less, and a high load is more than 100 Vusers.

The recommended file size for SIEBEL_SarmMaxFileSize is from 5000000 for low loads on the server to 25000000 for high loads on the server. If more than one Siebel log file is generated on the server every 10 seconds, you should increase the SIEBEL_SarmMaxFileSize.

Notes:

➤ For Siebel 7.7 the parameters differ slightly.

➤ Before running a load test, delete Siebel Diagnostics logs (***.sarm** files) from all servers involved in the load test.

To generate Siebel Server IDs:

On the Siebel application server, open a command window and run the command:

<Siebel bin directory>\srvrmgr /u <username> /p <password> /g <gateway server> /e <entrpr server> /c "list servers show SBLSRVR_NAME, SV_SRVRID"

where:

/u <username> is the server administrator username

/p <password> is the server administrator password

/g <gateway server> is the gateway server address

/e <entrpr server> is the enterprise server name

/c <command> is the execute a single command

This command generates a list of all the Siebel application servers and their IDs. Keep a record of the server IDs, since this information is required in the Siebel Server Configuration dialog box. For more information, see step 9 of "Setting Up the Siebel Diagnostics Module" on page 360.

Configuring Diagnostics where the Web Server is Inside a DMZ

If you are using an application server in the internal network and a Web (file) server in a DMZ (a "neutral zone" that separates an internal network from a public one that is used to prevent outside access to a company's private data), you must install the Mediator on the internal (over firewall) LAN, and enable SMB/CIFS communication from the internal machine to the file server in the DMZ. SMB/CIFS are the file sharing services that use the NBT (NetBIOS over TCP/IP) as the transport protocol.

To enable the NBT protocol between the client (over firewall machine) and the file server, use the following port configuration:

File Sharing Service	Port
SMB/CIFS over NBT	TCP 139 (SMB)
CIFS over TCP/IP (Direct SMB)	TCP 445

For example, configure the firewall settings as follows:

Service enabled: "nbsession" for TCP 139 connection.

Service enabled: "Microsoft-ds" for TCP 445 connection.

Note: CIFS over TCP 445 (direct SMB over TCP/IP) is optional with Windows 2000 and above (since it is a more secure way of communicating with the file server). To enable CIFS over TCP/IP, you must disable the NetBIOS over TCP/IP protocol using the operating system configuration.

Copying Files from the Siebel Application Server to the Mediator

After configuring the application server, you need to copy the files listed below from the Siebel Application server **bin** directory to either the <LR mediator installation>\bin directory, <Windows>\System32 directory, or any other directory in PATH on the Mediator machine:

For Siebel 7.53, copy the following files:

➤ sarmanalyzer.exe	➤ sslcshar.dll
➤ sslcver.dll	➤ sslcosa.dll
➤ sslcsym.dll	

For Siebel 7.7, copy the following files:

➤ sarmanalyzer.exe	➤ sslcosa.dll
➤ libarm.dll	➤ sslcosd.dll
➤ msvcp70.dll	➤ sslcrsa.dll
➤ msvcr70.dll	➤ sslcscr.dll
➤ sslcacln.dll	➤ sslcshar.dll
➤ sslccore.dll	➤ sslcsrd.dll
➤ sslcevt.dll	➤ sslcsym.dll
➤ sslcos.dll	➤ sslcver.dll

Setting Up the Siebel Diagnostics Module

To generate diagnostics data, you set up the Siebel Diagnostics module to communicate with the Mediator and define the servers that you want to monitor. You can then enable the diagnostics module and specify the sampling percentage of transaction data to include in the diagnostics graphs, as described in "Enabling LoadRunner Diagnostics" on page 351.

Notes:

➤ The settings that you configure are per scenario. All scripts in the scenario will run under the same diagnostics configuration.

➤ To ensure that valid diagnostics data is generated, manually define the transactions in the Vuser script rather than using automatic transactions. Make sure to disable the following options in the Run-Time Settings' **General : Miscellaneous** node: **Define each action as a transaction** and **Define each step as a transaction**.

To set up the Siebel Diagnostics module:

1 From the Controller menu, select **Diagnostics** > **Configuration**. The Diagnostics Distribution dialog box opens.

2 Make sure that **Enable the following diagnostics** is selected.

 3 In the **Offline Diagnostics** section, click the **Configure** button next to **Siebel Diagnostics (Max. Vuser Sampling: 10%)**.

 The Siebel Configuration dialog box opens.
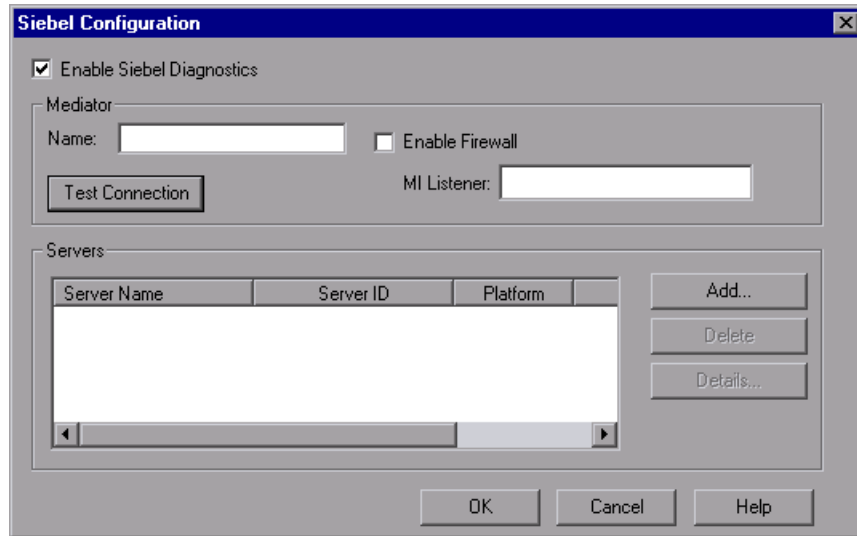
 4 Select **Enable Siebel Diagnostics**. This enables all the other fields in the dialog box.

 5 Enter the Mediator information as described in "Understanding the Siebel Configuration Dialog Box" on page 363.

 6 If you are monitoring over a firewall, select **Enable Firewall** and enter the name or IP address of the MI Listener machine. For more information, see Chapter 17, "Working with Firewalls in LoadRunner."

 7 To test the connection between the Controller and the Mediator, click **Test Connection**. The Siebel Diagnostics module attempts to connect to the Mediator.

 If the connection is not successful, see the Output window for more information. You can view the Output window by clicking the **Errors** link in the status bar.

 8 Click **Add** to add Siebel servers. The Siebel Server Configuration dialog box
 opens.
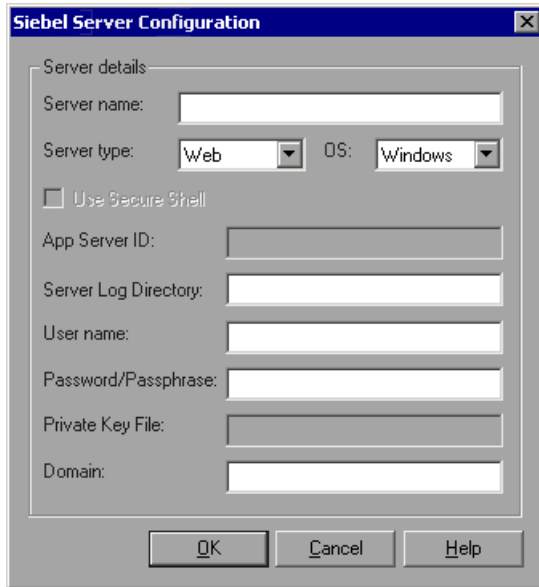
 9 Enter the Siebel server information as described in "Understanding the
 Siebel Server Configuration Dialog Box" on page 364, and then click **OK** to
 close the Siebel Server Configuration dialog box.

 10 In the Siebel Configuration dialog box, to show server information for a
 selected Siebel server, click **Details**.

 11 Click **OK** to close the Siebel Configuration dialog box. A check mark icon
 appears next to **Siebel Diagnostics (Max. Vuser Sampling: 10%)** in the
 Diagnostics Distribution dialog box, indicating that it is enabled.

 12 Click **OK** to close the Diagnostics Distribution dialog box.

Understanding the Siebel Configuration Dialog Box

You use the Siebel Configuration dialog box to define the Mediator, provide details of the monitored servers, and test the connection between the Controller and the Mediators.

➤ **Enable Siebel Diagnostics.** Select this option to enable Siebel Diagnostics and to configure the Siebel Diagnostics settings.

➤ **Mediator.**

   ➤ **Name.** Enter the name of the Mediator used to collect and process the Siebel diagnostics data. Only one Mediator is supported for each diagnostics module.

   Note: If you are using a Mediator that is over a firewall, enter the local machine key of the Mediator instead of the Mediator machine name.

   ➤ **Enable Firewall.** Select if the Mediator is over a firewall.

   ➤ **MI Listener.** Enter the name, full name, or IP address of the MI Listener machine if you are monitoring over a firewall.

   ➤ **Test Connection.** Tests the connections between the Siebel Diagnostics module and the Mediator.

   Note: The test connection does not check the connections to the Siebel servers.

➤ **Servers.**

   ➤ **Server Name.** The name of the Siebel server.

   ➤ **Server ID.** The Siebel server ID (for Siebel application servers only).

   ➤ **Platform.** The platform of the Siebel server.

   ➤ **Log Directory.** The Siebel server directory where Siebel log files (***.SARM**) are written.

➤ **Add.** Opens the Siebel Server Configuration dialog box enabling you to enter Siebel server information.

➤ **Delete.** Deletes a server from the server list.

➤ **Details.** Displays information for a selected server.

Understanding the Siebel Server Configuration Dialog Box

You use the Siebel Server Configuration dialog box to enter the Siebel server information.

➤ **Server Name.** Enter the name of the Siebel server.

➤ **Server Type.** Select the Siebel server type.

➤ **OS.** Select the Siebel server platform.

➤ **Use Secure Shell.** Select if you are working with a Secure Shell connection.

➤ **App Server ID.** Enter the Siebel server ID (required for Siebel application servers only). For information on generating a list of server IDs, see "Configuring Siebel Diagnostics on the Application and Web Servers" on page 356.

➤ **Server Log Directory.** Enter a location where the Siebel application saves the log files (*****.SARM). The log files can be saved in a shared log directory on the Siebel server or in a separate folder.

➤ **User Name.** Enter the user name of the machine where log files are stored.

Note: For Windows platforms, the user should have administrator privileges. See "Connecting to a Remote Windows Server" on page 347 for more information.
For UNIX platforms, see "Connecting to a Remote UNIX Server" on page 348.

➤ **Password/Passphrase.** Enter the user password or passphrase.

➤ **Private Key File.** Enter the name of the file where the Private Key is stored. This can be found on the Mediator. If you specify the file name only (without a path), the configuration will automatically look for the file in the Mediator's **<LoadRunner>\bin** directory.

➤ **Domain.** Enter the Siebel server domain.

21

Configuring Siebel DB Diagnostics

Siebel DB Diagnostics help you to rapidly identify and resolve database performance problems. You can view the SQLs for each transaction, identify the problematic SQL queries of each script, and identify at what point problems occurred. Siebel-Web Vusers support Siebel DB Diagnostics.

This chapter includes:

➤ Preparing the Script on page 365

➤ Synchronizing Clocks on page 366

➤ Enabling Server Logging on the Siebel Server on page 367

➤ Setting Up the Siebel DB Diagnostics Module on page 368

Preparing the Script

When preparing your script for collection of diagnostics data, it is recommended that you add think time at the end of each transaction using the ratio of one second per hour of testing.

To avoid session ID conflicts, make sure that the Vusers log off from the Siebel system at the end of each session.

Synchronizing Clocks

Synchronizing machine clocks before collecting diagnostics data ensures that the correlation of SQLs to transactions is correct.

Ensure that all the machines' clocks in the Siebel system are sychronized.

On a Windows Siebel server, synchronize the Siebel Gateway and load generators' clocks by running the following command from the load generator:

net time \ <Gateway name> /set /y

Replace <Gateway name> with the name of the Siebel Gateway.

On Unix Siebel servers you can synchronize the clocks in one of the following ways:

➤ Use the **date** command on the UNIX Siebel Gateway server to change the time manually, so it will be synchronized with the Load Generator's clock.

➤ Change the time on the load generator so that it will be synchronized with the UNIX Siebel Gateway server.

➤ Configure the time difference in Analysis. For more information see the chapter about Siebel DB Diagnostics Graphs in the *HP LoadRunner Analysis User's Guide*.

Enabling Server Logging on the Siebel Server

Before you set up the Siebel DB Diagnostics module on the Controller, you must configure the Siebel server to create the database log files.

To enable logging on the Siebel server:

1 On the Siebel server, open a command window and run the command:

<Siebel bin directory>\srvrmgr /g <gateway server> /s <Siebel server> /e <enterprise server name> /u <username> /p <password>

where:

/u <username> is the server administrator username

/p <password> is the server administrator password

/g <gateway server> is the gateway server address

/e <entrpr server> is the enterprise server name

/c <command> is the execute a single command

2 Enter the following commands:

change evtloglvl ObjMgrsqllog=4 for comp <component name>

evtloglvl EventContext=3 for comp <component name>

evtloglvl ObjMgrSessionInfo =3 for comp <component name>

For example, for the Call Center component, enter sccobjmgr_enu as the component name, as follows:

change evtloglvl ObjMgrsqllog=4 for comp sccobjmgr_enu

To disable logging on the Siebel server:

On the Siebel server, enter the following commands:

change evtloglvl ObjMgrsqllog=0 for comp <component name>

change evtloglvl EventContext=0 for comp <component name>

change evtloglvl ObjMgrSessionInfo =0 for comp <component name>

Note: Before running a load test, delete log files from all servers involved in the load test.

Setting Up the Siebel DB Diagnostics Module

To generate diagnostics data, you set up the Siebel DB Diagnostics module to communicate with the Mediator and define the servers that you want to monitor. You can then enable the diagnostics module and specify the sampling percentage of transaction data to include in the diagnostics graphs, as described in "Enabling LoadRunner Diagnostics" on page 351.

Notes:

➤ The settings that you configure are per scenario. All scripts in the scenario will run under the same diagnostics configuration.

➤ To ensure that valid diagnostics data is generated, manually define the transactions in the Vuser script rather than using automatic transactions. Make sure to disable the following options in the Run-Time Settings' **General : Miscellaneous** node: **Define each action as a transaction** and **Define each step as a transaction**.

To set up the Siebel DB Diagnostics module:

1 From the Controller menu, select **Diagnostics > Configuration**. The Diagnostics Distribution dialog box opens.

2 Make sure that **Enable the following diagnostics** is selected.

3 In the **Offline Diagnostics** section, click the **Configure** button next to **Siebel DB Diagnostics (Max. Vuser Sampling: 10%)**. The Siebel DB Configuration dialog box opens.

4 Select **Enable Siebel DB Diagnostics**. This enables all the other fields in the dialog box.

5 Enter the Mediator information as described in "Understanding the Siebel DB Configuration Dialog Box" on page 371.

6 If you are monitoring over a firewall, select **Enable Firewall** and enter the name or IP address of the MI Listener machine. For more information, see For more information, see Chapter 17, "Working with Firewalls in LoadRunner."

7 To test the connection between the Controller and the Mediator, click **Test Connection**. The Siebel DB Diagnostics module attempts to connect to the Mediator.

If the connection is not successful, see the Output window for more information. You can view the Output window by clicking the **Errors** link in the status bar.

8 To add Siebel DB servers, click **Add**. The Siebel DB Server Configuration dialog box opens.

9 Enter the Siebel DB server information as described in "Understanding the Siebel DB Server Configuration Dialog Box" on page 372, and then click **OK** to close the Siebel DB Server Configuration dialog box.

10 In the Siebel DB Configuration dialog box, click **Details** to show server information for a selected Siebel server.

11 Click **OK** to close the Siebel DB Configuration dialog box.

In the Diagnostics Distribution dialog box, a check mark icon appears next to **Siebel DB Diagnostics (Max. Vuser Sampling: 10%)**, indicating that it is enabled.

12 Click **OK** to close the Diagnostics Distribution dialog box.

Understanding the Siebel DB Configuration Dialog Box

You use the Siebel DB Configuration dialog box to define the Mediator, provide details of monitored servers, and test the connection between the Controller and the Mediator machines.

➤ **Enable Siebel DB Diagnostics.** Select this option to enable Siebel DB Diagnostics and to configure the Siebel DB Diagnostics settings.

➤ **Mediator.**

➤ **Name.** Enter the name of the Mediator used to collect and process the Siebel DB Diagnostics data. Only one Mediator is supported for each diagnostics module.

Note: It may take a long time to copy the Siebel DB diagnostics files from the application server to the Mediator, and then from the Mediator to the results directory. During the first copying stage, the Summary Data Processing window is displayed. To minimize copying time in the second stage (if you are not working over a firewall), we recommend using localhost as the Mediator machine. This reduces the time it takes to copy the Siebel DB diagnostics files to the results directory, since the diagnostics files are already on the Controller machine.

If you are using a Mediator that is over a firewall, enter the local machine key of the Mediator instead of the Mediator machine name.

➤ **Enable Firewall.** Select if the Mediator is over a firewall.

➤ **MI Listener.** Enter the name, full name, or IP address of the MI Listener machine if you are monitoring over a firewall.

➤ **Test Connection.** Tests the connections between the Siebel DB Diagnostics module and the Mediator machine.

Note: The test connection does not check the connections to the Siebel servers.

➤ **Servers.**

➤ **Server.** The name of the Siebel server.

➤ **Platform.** The platform of the Siebel server.

➤ **Log Directory.** The directory where Siebel log files are written.

➤ **Add.** Opens the Siebel DB Server Configuration dialog box enabling you to enter Siebel server information.

➤ **Delete.** Deletes a server from the server list.

➤ **Details.** Displays information for a selected server.

Understanding the Siebel DB Server Configuration Dialog Box

You use the Siebel DB Server Configuration dialog box to enter the Siebel server information.

➤ **Server Name.** Enter the name of the Siebel server.

➤ **Server Platform.** Select the Siebel server platform.

➤ **Use Secure Shell.** Select if you are working with a Secure Shell connection.

➤ **Server Log Directory.** Enter a location where the Siebel application saves the log files. The log files can be saved in a shared log directory on the Siebel server or in a separate folder.

➤ **User Name.** Enter the user name of the machine where log files are stored.

Note: For Windows platforms, the user should have administrator privileges. See "Connecting to a Remote Windows Server" on page 347 for more information.
For UNIX platforms, see "Connecting to a Remote UNIX Server" on page 348.

➤ **Password/Passphrase.** Enter the user password or passphrase.

➤ **Private Key File.** Enter the name of the file where the Private Key is stored. This can be found on the Mediator machine. If you specify the file name only (without a path), the configuration will automatically look for the file in the Mediator's <**LoadRunner**>**bin** directory.

➤ **Domain.** Enter the Siebel server domain.

22

Configuring Oracle 11i Diagnostics

Oracle 11i Diagnostics helps pinpoint performance problems on Oracle NCA systems. The diagnostics information drills down from the transaction, to the SQL statements and the SQL stages of each statement. Oracle NCA Vusers support Oracle 11i Diagnostics.

This chapter includes:

➤ Enabling Server Logging On the Oracle Server on page 373

➤ Selecting the Oracle NCA Application Version on page 375

➤ Setting Up the Oracle 11i Diagnostics Module on page 376

Enabling Server Logging On the Oracle Server

To enable server logging on the Oracle server, verify that the trace diagnostics are enabled and set the trace file size to unlimited. By default, trace diagnostics are enabled on the Oracle server during installation. In addition, to help LoadRunner deal with the Oracle application diagnostics password, you can either set the diagnostics password in the Vuser script or disable the password request on the application server.

To check that trace diagnostics are enabled:

1 Log on to the Oracle application server with administrator privileges, and select the module you want in the Oracle Application. The Responsibilities dialog box opens.

2 Select **System Administrator** and click **OK**.

3 In the **Functions** tab, select **Profile** > **System** and click **Open**. The System Profile Values dialog box opens.

4 In the **Display** section, select **Site** and **Profiles with No Values**, enter %Diagnostics% in the **Profiles** field, and then click **Find**.

5 If any diagnostics profiles are disabled (denoted by a "Yes" in the **Site** column), change the setting to "No".

6 Save your settings.

To set the trace file size to unlimited:

For Oracle 9i:

On the Oracle server, run the following command in the SQL editor:

Alter system set max_dump_file_size=UNLIMITED scope=both;

For Oracle 8i:

1 On the Oracle server, run the following command in the SQL editor:

Alter system set max_dump_file_size=2048000;

2 Edit the **init*.ora** file on **$ORACLE_HOME\admin\<sid>\pfile\init<sid>.ora**. Find the line of the parameter, change its value, and then save the file.

Note: Verify that you have enough disk space on the database server since these trace file can be very large.

To set the diagnostics password in the Vuser script:

In VuGen, add the **nca_set_diagnostics_password(<password>)** function to your script and select a password.

Note: The **nca_set_diagnostics_password** function must come after the **nca_connect_server** function.

To disable the diagnostics password request on the application server:

1 Log on to the Oracle application server with administrator privileges, and select the module you want in the Oracle Application. The Responsibilities dialog box opens.

2 Select **System Administrator** and click **OK**.

3 In the **Functions** tab, select **Profile** > **System** and click **Open**. The System Profile Values dialog box opens.

4 In the **Display** section, select **User**, and enter the required user name. In the **Profile** field, enter %Utilities:Diagnostics% and click **Find**. The Utilities:Diagnostics profile values are displayed.

5 In the **User** column of the Utilities:Diagnostics profile, set the value to **Yes**.

6 Save your settings.

Selecting the Oracle NCA Application Version

The Oracle 11i diagnostics module supports Oracle NCA versions 11.5.0 and later. Enter the version of your Oracle application server in VuGen's run-time settings to enable the built-in trace mechanism. To check the version of your Oracle server, log in to the Oracle server and select **Help** > **About Oracle**. The version of your Oracle server is displayed in the Oracle Application field.

To enter your Oracle application version:

Open the script in VuGen and select **Vuser** > **Run-Time Settings**. In the **Oracle NCA: Client Emulation** node, select the version of Oracle NCA that you are using in the **Diagnostics** > **Application Version** field.

Note: If the Oracle 11i trace cannot be enabled using the built-in mechanism, you can enable it manually in the Vuser script using the **nca_set_custom_dbtrace** and **nca_set_dbtrace_file_index** functions. This may occur if you are using a custom application that does not have a standard UI.

Note: Before running a load test, delete trace log files from all servers involved in the load test.

No real or other virtual users should work on the Oracle server while the diagnostics module is running, as this may affect diagnostics results.

Setting Up the Oracle 11i Diagnostics Module

To generate diagnostics data, you set up the Oracle 11i Diagnostics module to communicate with the Mediator machine and define the servers that you want to monitor. You can then enable the diagnostics module and specify the sampling percentage of transaction data to include in the diagnostics graphs, as described in "Enabling LoadRunner Diagnostics" on page 351.

Notes:

➤ The settings that you configure are per scenario. All scripts in the scenario will run under the same diagnostics configuration.

➤ To ensure that valid diagnostics data is generated, manually define the transactions in the Vuser script rather than using automatic transactions. Make sure to disable the following options in the Run-Time Settings' **General : Miscellaneous** node: **Define each action as a transaction** and **Define each step as a transaction**.

To set up the Oracle 11i Diagnostics module:

1 From the Controller menu, select **Diagnostics** > **Configuration**. The Diagnostics Distribution dialog box opens.

2 Make sure that **Enable the following diagnostics** is selected.

3 In the **Offline Diagnostics** section, click the **Configure** button next to **Oracle 11i Diagnostics (Max. Vuser Sampling: 5%)**.

The Oracle 11i Configuration dialog box opens.
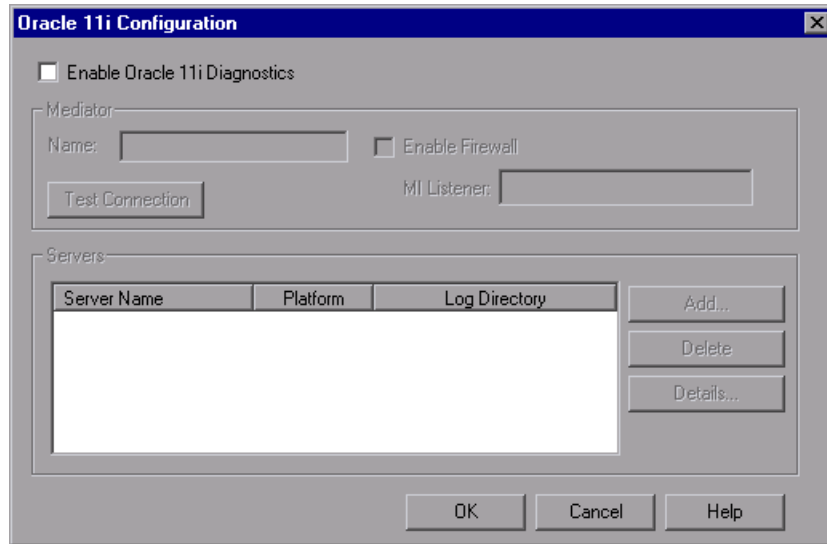
4 Select **Enable Oracle 11i Diagnostics**. This enables all the other fields in the dialog box.

5 Enter the Mediator information as described in "Understanding the Oracle 11i Configuration Dialog Box" on page 379.

6 If you are monitoring over a firewall, select **Enable Firewall** and enter the name or IP address of the MI Listener machine. For more information, see For more information, see Chapter 17, "Working with Firewalls in LoadRunner."

7 To test the connection between the Controller and the Mediator, click **Test Connection**. The Oracle 11i Diagnostics module attempts to connect to the Mediator.

If the connection is not successful, see the Output window for more information. You can view the Output window by clicking the **Errors** link in the status bar.

  8 Click **Add** to add Oracle servers. The Oracle 11i Server Configuration dialog box opens.
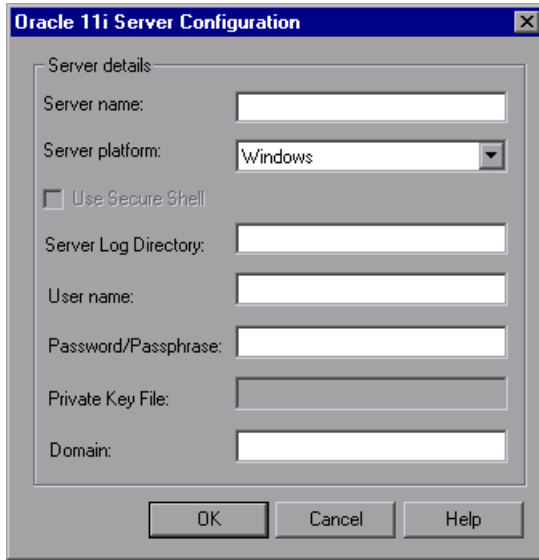
  9 Enter the Oracle server information as described in "Understanding the Oracle 11i Server Configuration Dialog Box" on page 380, and then click **OK**.

  10 In the Oracle 11i Configuration dialog box, click **Details** to show server information for a selected Oracle server.

  11 Click **OK** to close the Oracle 11i Configuration dialog box.

  In the Diagnostics Distribution dialog box, a check mark icon appears next to **Oracle 11i Diagnostics (Max. Vuser Sampling: 5%)**, indicating that it is enabled.

  12 Click **OK** to close the Diagnostics Distribution dialog box.

Understanding the Oracle 11i Configuration Dialog Box

You use the Oracle 11i Configuration dialog box to define the Mediator machine, provide details of monitored servers, and test the connection between the Controller and the Mediator machines.

➤ **Enable Oracle 11i Diagnostics**. Select this option to enable Oracle 11i Diagnostics and to configure the Oracle 11i Diagnostics settings.

➤ **Mediator.**

  ➤ **Name.** Enter the name of the Mediator machine used to collect and process the Oracle 11i diagnostics data. Only one mediator machine is supported for each diagnostics module.

  Note: If you are using a mediator that is over a firewall, enter the local machine key of the Mediator instead of the Mediator machine name.

  ➤ **Enable Firewall.** Select if the Mediator is over a firewall.

  ➤ **MI Listener.** Enter the name, full name, or IP address of the MI Listener machine if you are monitoring over a firewall.

  ➤ **Test Connection.** Tests the connections between the Oracle 11i Diagnostics module and the Mediator machine.

  Note: The test connection does not check the connections to the Oracle servers.

➤ **Servers**

  ➤ **Server.** The name of the Oracle server.

  ➤ **Platform.** The platform of the Oracle server.

  ➤ **Log Directory.** The directory where Oracle trace files (***.trc**) are written.

➤ **Add.** Opens the Oracle 11i Server Configuration dialog box where you enter Oracle server information.

➤ **Delete.** Deletes a server from the server list.

➤ **Details.** Displays information for a selected server.

Understanding the Oracle 11i Server Configuration Dialog Box

You use the Oracle 11i Server Configuration dialog box to enter the Oracle 11i server information.

➤ **Server Name.** Enter the name of the Oracle server.

➤ **Server Platform.** Select the Oracle server platform.

➤ **Use Secure Shell.** Select if you are working with a Secure Shell connection.

➤ **Server Log Directory.** Enter a location where the Oracle application saves the trace files. The trace files can be saved in a shared directory on the Oracle server or in a separate folder.

➤ **User Name.** Enter the user name of the machine where trace files are stored.

Note: For Windows platforms, the user should have administrator privileges. See "Connecting to a Remote Windows Server" on page 347 for more information.
For UNIX platforms, see "Connecting to a Remote UNIX Server" on page 348.

➤ **Password/Passphrase.** Enter the user password or passphrase.

➤ **Private Key File.** Enter the name of the file where the Private Key is stored. This can be found on the Mediator machine. If you specify the file name only (without a path) the configuration will automatically look for the file in the Mediator's <**LoadRunner**>**bin** directory.

➤ **Domain.** Enter the Oracle server domain.

23

Configuring SAP Diagnostics

SAP Diagnostics enables a you to pinpoint the root cause of a certain problem (i.e. DBA, Network, WAS, Application, OS/HW) quickly and easily, Once you have specific information, you can consult with the relevant expert only, without having to present the problem to a whole team of people.

This chapter includes:

➤ SAP Diagnostics Supported Environments on page 381

➤ SAP Diagnostics Overview on page 382

➤ Setting Up the SAP Diagnostics Module on page 382

SAP Diagnostics Supported Environments

The following table outlines the supported versions and required Kernel patches for the SAP Application Server and the SAPGUI Client:

	Supported Version	Required Kernel Patch
SAP Application Server	4.6C; 4.6D	Kernel Patch 1984 (released on 11/01/05, SAP note 0451251)
	4.7 and higher	No patch required.
SAPGUI Client	SAPGUI for Windows 6.20	Minimal patch level: 48
	SAPGUI for Windows 6.40	Minimal patch level: 2

SAP Diagnostics Overview

Server-side Data Collection

SAP Diagnostics is based on SAP Distributed Statistical Records (DSR). For each dialog step performed on application server, a statistical record is generated, which includes information such as response time components, database statistics, RFC times etc.

The Mediator is responsible for collecting diagnostics data from the server during a scenario run.

Vuser Coloring

SAP Diagnostics works for replay of the SAPGUI protocol. The user sets the percentage of colored Vusers in the Configuration dialog (up to 100% allowed). Coloring of SAPGUI users creates no overhead on the Application server.

Setting Up the SAP Diagnostics Module

To generate diagnostics data, you set up the SAP Diagnostics module to communicate with the Mediator machine and define the servers that you want to monitor. You can then enable the diagnostics module and specify the sampling percentage of transaction data to include in the diagnostics graphs, as described in "Enabling LoadRunner Diagnostics" on page 351.

Notes:

➤ The settings that you configure are per scenario. All scripts in the scenario will run under the same diagnostics configuration.

➤ To ensure that valid diagnostics data is generated, manually define the transactions in the Vuser script rather than using automatic transactions. Make sure to disable the following options in the Run-Time Settings' **General : Miscellaneous** node: **Define each action as a transaction** and **Define each step as a transaction**.

To set up the SAP Diagnostics module:

1 From the Controller menu, select **Diagnostics > Configuration**. The Diagnostics Distribution dialog box opens.

2 Make sure that **Enable the following diagnostics** is selected.

3 In the **Offline Diagnostics** section, click the **Configure** button next to **SAP Diagnostics (Max. Vuser Sampling: 100%)**.. The SAP Configuration dialog box opens.

4 Select **Enable SAP Diagnostics**. This enables all the other fields in the dialog box.

5 Enter the Mediator information as described in "Understanding the SAP Configuration Dialog Box" on page 384.

6 If you are monitoring over a firewall, select **Enable Firewall** and enter the name or IP address of the MI Listener machine. For more information, see Chapter 17, "Working with Firewalls in LoadRunner."

7 Enter the SAP and Client server properties in the **Properties** section.

8 Click **Validate** to ensure that the server is connected properly.

When you click **Validate**, the Controller produces a report of all the servers that are available for diagnostics through the Server Host.

9 Click **OK** to close the SAP Configuration dialog box.

In the Diagnostics Distribution dialog box, a check mark icon appears next to **SAP Diagnostics (Max. Vuser Sampling: 100%)** , indicating that it is enabled.

10 Click **OK** to close the Diagnostics Distribution dialog box.

Understanding the SAP Configuration Dialog Box

You use the SAP Configuration dialog box to define the Mediator machine, provide details of monitored servers, and validate that the Mediator can collect diagnostics data.

➤ **Enable SAP Diagnostics.** Select this option to enable SAP Diagnostics and to configure the SAP Diagnostics settings.

➤ **Mediator.**

  ➤ **Name.** Enter the name of the Mediator machine used to collect and process the SAP diagnostics data. Only one mediator machine is supported for each diagnostics module.

   Note: If you are using a mediator that is over a firewall, enter the local machine key of the Mediator instead of the Mediator machine name.

  ➤ **Enable Firewall.** Select if the Mediator is over a firewall.

  ➤ **MI Listener.** Enter the name, full name, or IP address of the MI Listener machine if you are monitoring over a firewall.

➤ **Properties.**

  ➤ **SAP Server properties.**

    ➤ **Application server.** The name of the SAP server.

    ➤ **Router string.** Enter the system router string of the SAP server (optional).

    ➤ **System number.** Enter the system number of the SAP server.

  ➤ **Client properties.** The Mediator uses these details to connect to the SAP server and collect diagnostics data.

    ➤ **User name.** The user's unique name for logging on to the SAP Server.

➤ **Password.** The user's password for logging on to the SAP Server.

➤ **Client number.** The client number of the selected user.

➤ **Validate.** Click to validate connection to the SAP server.

Part VI

Monitoring a Scenario

24

Online Monitoring

You can monitor scenario execution using the LoadRunner online monitors.

This chapter describes the online monitor user interface. The specific monitors are discussed in the *HP LoadRunner Monitor Reference*.

This chapter includes:

About Online Monitoring

LoadRunner enables you to view data generated during scenario execution using the online monitors. You specify the machines that the Controller will monitor during a scenario execution, and view the data collected by the monitors, using the LoadRunner online graphs.

A primary factor in a transaction's response time is its resource usage. By monitoring resources during a scenario run, you can determine why a bottleneck occurred on a particular machine. LoadRunner's server resource monitors let you keep track of resources used during a scenario. LoadRunner displays the selected resource monitors in real time during test execution. You can select the server resource measurements to monitor both before and during the scenario.

Setting Up the Monitoring Environment

Before monitoring a scenario, you need to set up and configure the LoadRunner monitoring components. Each monitor has different configuration requirements which are explained in the specific monitoring chapters in the *HP LoadRunner Monitor Reference*. The diagram below shows the LoadRunner monitoring process.
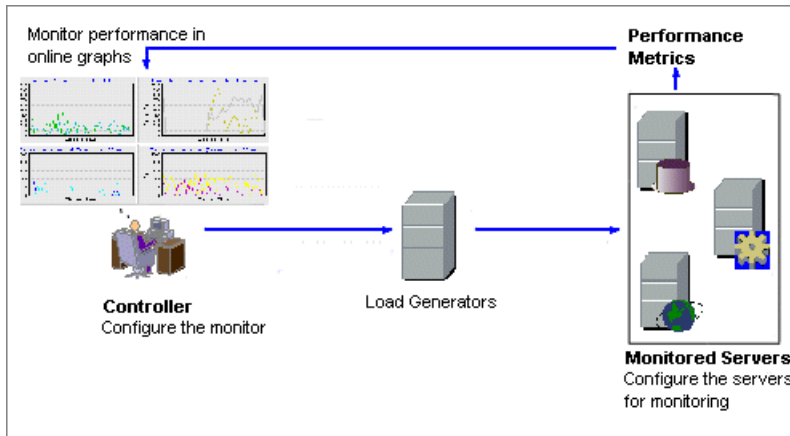
Before monitoring a server, perform the following steps:

➤ Configure the monitoring environment on the server machine (if necessary).

➤ Configure the monitor on the Controller machine.

Configuring the Monitoring Environment on the Server Machine

To use the following monitors, you must first install or configure monitoring components on the server machine:

➤ COM+

➤ Citrix

➤ DB2

➤ IBM WebSphere MQ

➤ iPlanet (NAS)

➤ J2EE

➤ J2EE & .NET Diagnostics

➤ Network Delay

➤ Oracle

➤ PeopleSoft (Tuxedo)

➤ SAP CCMS

➤ SAP Portal

➤ SAPGUI

➤ Siebel Server Manager

➤ Siebel Web Server

➤ SiteScope

➤ Sybase

➤ Tuxedo

➤ UNIX

➤ WebLogic (JMX)

➤ WebSphere (EPM)

➤ WebSphere

➤ WebSphere Application Server

Configuring the Monitor on the Controller Machine

To obtain performance data for a monitor, you need to enable the monitor (from the Controller), and indicate which statistics and measurements you want to monitor. You select these counters using the monitor's Add Measurements dialog box.

If you are configuring a SiteScope monitor you need to configure the remote machine. For details, see "Configuring the Remote Machine for SiteScope Monitors" on page 402.

For more information on setting up the monitoring environment and configuring a monitor, see the specific monitoring chapter in the *HP LoadRunner Monitor Reference*.

Monitor Types

The online monitors are divided into the following categories:

➤ **Run-Time Monitors.** Display the number and status of Vusers participating in the scenario, as well as the number and types of errors that the Vusers generate. For more information, see the chapter "Run-Time and Transaction Monitoring" in the *HP LoadRunner Monitor Reference*.

➤ **Transaction Monitors.** Display the transaction rate and response time during scenario execution. For more information, see the chapter "Run-Time and Transaction Monitoring" in the *HP LoadRunner Monitor Reference*.

➤ **Web Resource Monitors.** Provide information about the number of Web connections, throughput volume, HTTP responses, server retries, and pages downloaded to the Web servers during the scenario. For more information, see the "Web Resource Monitoring" section in the *HP LoadRunner Monitor Reference*.

➤ **System Resource Monitors.** Measure the Windows, UNIX, Tuxedo, SNMP, Antara FlameThrower, and SiteScope resources used during a scenario. For more information, see the "System Resource Monitoring" section in the *HP LoadRunner Monitor Reference*.

➤ **Network Delay Monitor.** Displays information about the network delays on your system. For more information, see the "Network Monitoring" section in the *HP LoadRunner Monitor Reference*.

➤ **Firewall Monitor.** Measures statistics of the firewall servers during the scenario. For more information, see the "Firewall Server Performance Monitoring" section in the *HP LoadRunner Monitor Reference*.

➤ **Web Server Resource Monitors.** Measure statistics of the Apache, Microsoft IIS, iPlanet (SNMP), and iPlanet/Netscape Web servers during the scenario. For more information, see the "Web Server Resource Monitoring" section in the *HP LoadRunner Monitor Reference*.

➤ **Web Application Server Resource Monitors.** Measure statistics of the Ariba, ATG Dynamo, BroadVision, ColdFusion, Fujitsu INTERSTAGE, iPlanet (NAS), Microsoft ASP, Oracle9iAS HTTP, SilverStream, WebLogic (SNMP), WebLogic (JMX), and WebSphere application servers during the scenario. For more information, see the "Web Application Server Resource Monitoring" section in the *HP LoadRunner Monitor Reference*.

➤ **Database Server Resource Monitors.** Measure statistics of the SQL server, Oracle, Sybase, and DB2 databases during the scenario. For more information, see the "Database Resource Monitoring" section in the *HP LoadRunner Monitor Reference*.

➤ **Streaming Media Monitors.** Measure statistics of the Windows Media Server and RealPlayer audio/video servers, as well as the RealPlayer and Media Player client during the scenario. For more information, see the "Streaming Media Monitoring" section in the *HP LoadRunner Monitor Reference*.

➤ **ERP/CRM Server Resource Monitors.** Measure statistics of the SAP Portal, SAP CCMS, SAPGUI, Siebel Server Manager, Siebel Web Server, and PeopleSoft (Tuxedo) servers during the scenario. For more information, see the "ERP/CRM Server Resource Monitoring"section in the *HP LoadRunner Monitor Reference*.

➤ **Java Performance Monitor.** Measures statistics of Java 2 Platform, Enterprise Edition (J2EE) objects, and Java-based applications, using J2EE machines. For more information, see the "Java Performance Monitoring" section in the *HP LoadRunner Monitor Reference*.

➤ **J2EE & .NET Diagnostics Monitors.** Provide information to trace, time, and troubleshoot individual transactions through J2EE & .NET Web, application, and database servers. For more information, see the *HP Diagnostics User's Guide*.

➤ **Application Component Monitor.** Measures statistics of the Microsoft COM+ server during a scenario run. For more information, see the "Application Component Monitoring" section in the *HP LoadRunner Monitor Reference*.

➤ **Application Deployment Solutions Monitor.** Measures statistics of the Citrix MetaFrame XP and 1.8 servers during a scenario run. For more information, see the "Application Deployment Solutions" section in the *HP LoadRunner Monitor Reference*.

➤ **Middleware Performance Monitors.** Measure statistics of the Tuxedo and IBM WebSphere MQ servers during a scenario run. For more information, see the "Middleware Performance Monitoring" section in the *HP LoadRunner Monitor Reference*.

➤ **Infrastructure Resources Monitor.** Measures statistics of the network client data points during a scenario run. For more information, see the "Infrastructure Resources Monitoring" section in the *HP LoadRunner Monitor Reference*.

For information on configuring graph settings and measurements, and exporting graph data, see Chapter 25, "Configuring Online Graphs."

All of the monitors allow you to view a summary of the collected data at the conclusion of the scenario. Using Analysis, you can generate a graph for any of the monitors. For more information, see the *HP LoadRunner Analysis User's Guide*.

Note: Application Component monitors are available only in the Controller.

For a detailed list of LoadRunner's monitors, see the HP Software Web site http://www.mercury.com/us/products/performance-center/loadrunner/monitors/.

Choosing Monitors and Measurements in the Controller

You specify the machines and measurements that the Controller will monitor during a scenario execution, using the monitored server machine dialog box.

To monitor a machine's resources:

 1 Open the graph that you want to monitor in the graph view area and select **Monitors** > **Add Measurements**, or right-click the graph and select **Add Measurements**. The monitored server machine dialog box opens.
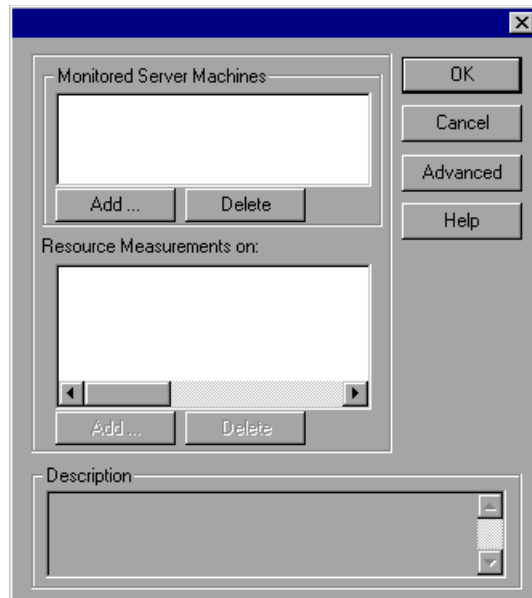
 2 Some monitors are native LoadRunner monitors (by default), but can also monitor through the SiteScope monitor engine.

If you want to monitor a server through the SiteScope monitor engine, click **Advanced**. The Choose Monitor Engine dialog box opens.

Select **SiteScope**, and click **OK**.

For more information, see the relevant monitoring section.

3 In the **Monitored Server Machines** section, click **Add.** The Add Machine dialog box opens.
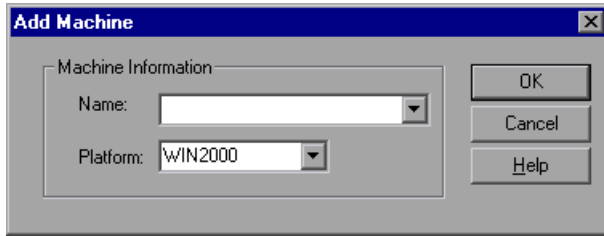
For monitors that use SiteScope to monitor the server, the **SiteScope Server Information** section is also displayed in the Add Machine dialog box:
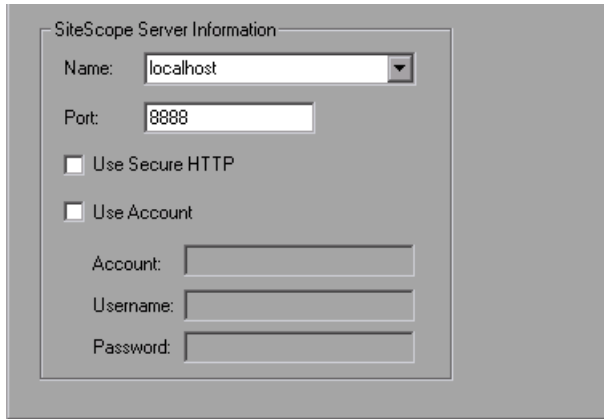
4 Enter the server name or IP address of the machine you want to monitor, select the platform on which the machine runs.

For Sitescope monitors enter the name and port number of the SiteScope server, and specify whether you are using a Secure HTTP connection. To use an account, fill in the relevant account information. For more information, see "Understanding the Add Machine Dialog Box" on page 401.

Click **OK**.

The machine is displayed in the **Monitored Server Machines** box of the monitored server machine dialog box.
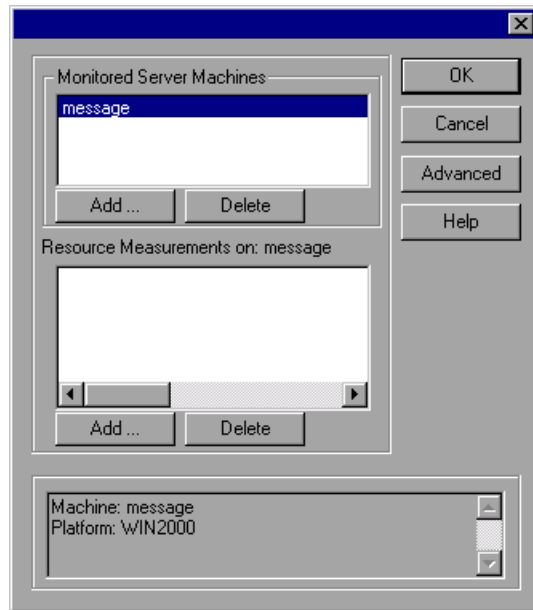
5 Select the server machine that you want to monitor and, under the **Resource Measurements on:** *<machine>* box, click **Add**.

The monitored server's Add Measurements dialog box opens, displaying the available measurements.
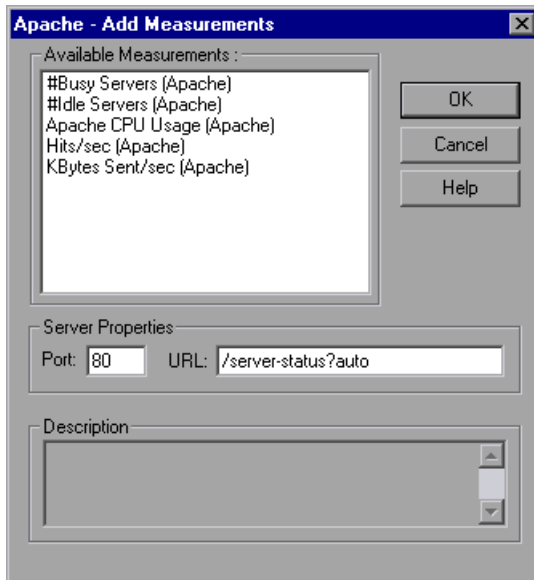
Note:

➤ The Add Measurements dialog box is different for each monitor. See the relevant monitoring section for specific add measurement instructions.

➤ When you add a new SiteScope monitors, you need to configure the remote machine. For details, see "Configuring the Remote Machine for SiteScope Monitors" on page 402.

➤ Some monitors need to be configured before adding the measurements. See the relevant monitoring section for more specific configuration instructions.

6 Select the required measurements. You can select multiple measurements using the CTRL key.

7 Click **OK.** The Add Measurements window closes, and the selected measurements are displayed in the **Resource Measurement on:** *<machine>* box of the monitored server machine dialog box.
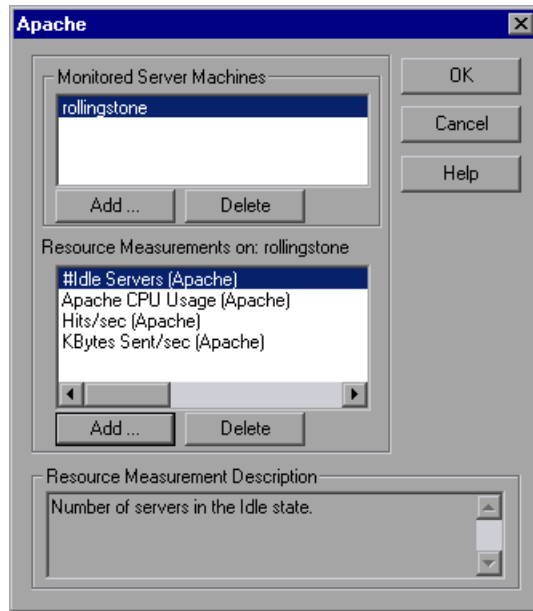
8 Click **OK** in the monitored server machine dialog box to activate the monitor.

Understanding the Monitored Server Machine Dialog Box

➤ **Monitored Server Machines.** The machines whose resources are being monitored.

   ➤ **Add.** Displays the Add Machine dialog box, which adds the machine that you want to monitor to the existing list.

   ➤ **Delete.** Removes the selected machine from the list.

Note: In some cases, you can (or must) specify the server you want to monitor using other formats. See the relevant monitoring section in the *HP LoadRunner Monitor Reference* for more information.

➤ **Resource Measurements on <machine name>.** Displays the resource measurements being monitored on the selected machine.

   ➤ **Add.** Opens a dialog box that lets you create a list of resources to monitor on the selected machine.

   ➤ **Delete.** Removes the selected resource measurement from the list.

➤ **Advanced.** Opens the Choose Monitor Engine dialog box, for selecting native LoadRunner or SiteScope monitoring.

   ➤ To monitor a server through LoadRunner, select **LoadRunner native monitors**.

   ➤ To monitor a server through SiteScope, select **SiteScope**.

Note: This button is enabled only for those monitors that can be either a SiteScope monitor or a native LoadRunner monitor.

➤ **Description.** Displays a description of the selected resource measurement.

Note: For information about setting up a specific server monitor before configuring its measurements, see the relevant monitoring section in the *HP LoadRunner Monitor Reference*.

Understanding the Add Machine Dialog Box

Adds the machine that you want to monitor to the Monitored Server Machines list.

➤ **Monitored Machine Information.**

➤ **Name.** Enter the name or IP address of the machine that you want to monitor.

➤ **Platform.** Enter the platform of the machine you want to monitor.

➤ **SiteScope Server Information.** For monitors that use SiteScope, enter the following SiteScope server information:

➤ **Name.** Enter the name of the SiteScope server.

➤ **Port.** Enter the SiteScope port (default:8888).

➤ **Use Secure HTTP.** Select this to use a Secure HTTP connection.

➤ **Use Account.** Select this option to use a specific SiteScope user account. Enter the following account details:

➤ **Account.** The SiteScope account name or number.

➤ **Username.** The username defined to log in to the SiteScope account.

➤ **Password.** The password defined to log in to the SiteScope account.

Configuring the Remote Machine for SiteScope Monitors

You configure the remote machine according to the platform your machine is running on.

Configuring the NT Remote Machine

For machine on a NT platform, when you add measurements to monitor for the first time, the Configuring NT Remote Machine dialog box opens.
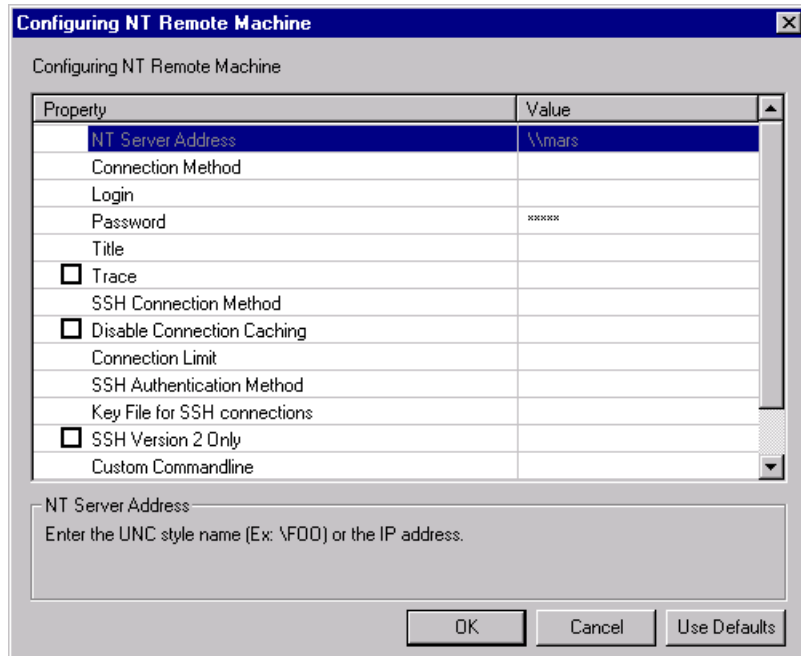
Select settings for the remote machine as described in "Understanding the Configuring NT Remote Machine Dialog Box" on page 403.

Configuring the UNIX Remote Machine

For machine on a UNIX platform, when you add measurements to monitor for the first time, the Configuring Unix Remote Machine dialog box opens.

Select settings for the remote machine as described in "Understanding the Configuring Unix Remote Machine Dialog Box" on page 406.

Understanding the Configuring NT Remote Machine Dialog Box

You configure the remote NT machine from the Configuring NT Remote Machine dialog box.

➤ **NT Server Address.** The IP address or UNC style name of the NT server you wish to monitor. An IP hostname will also work provided that the SiteScope server has a way to resolve this common name into an IP address (for example, by the use of a hosts file, DNS, or WINS/DNS integration).

➤ **Connection Method.** SiteScope can use one of two connection types for monitoring remote NT server resources. These are:

➤ **NetBIOS.** The default server-to-server communication protocol for Windows NT and 2000 networks.

➤ **SSH.** Secure Shell, a more secure communication protocol that can be installed on Windows NT/2000 based networks. This connection method normally requires installing SSH libraries on each server to which you want to connect.

➤ **Login.** The login for the remote server. If the server is within the same domain as the SiteScope machine, include the domain name in front of the user login name. For example: domainname\user. If you are using a local machine login account for machines within or outside the domain, include the machine name in front of the user login name. For example: machinename\user.

➤ **Password.** The password for the remote server or the passphrase for the SSH key file.

Note: When using SSH authentication with public/private key based authentication enter the passphrase for the identity file here.

➤ **Title.** (Optional) A name by which the remote machine should be known. This name will appear in the drop-down list.

➤ **Trace.** Check this box to have trace messages to and from the subject server recorded to the SiteScope RunMonitor.log file.

➤ **SSH Connection Method.** The method to use for this connection. The currently supported methods are:

➤ **Internal Java Libraries.** Connect using the Java SSH client integrated with SiteScope.

➤ **Plink.** Connect using an external SSH client. On NT, SiteScope ships with Plink. On UNIX or Linux SiteScope will use an installed client such as OpenSSH.

➤ **Disable Connection Caching.** Check this option to turn off connection caching for this remote. By default SiteScope caches open connections.

➤ **Connection Limit.** Controls the number of open connections that SiteScope will allow for this remote. If you have a large number of monitors configured to use this connection then set this number high enough to relieve the potential bottleneck.

Note: This setting does not effect the running of tests for a remote, tests will always create a new connection.

➤ **SSH Authentication Method.** The authentication method to use for SSH connections. The currently supported methods are:

➤ **Password.** Authenticate using a password.

➤ **Key File.** Authenticate using public/private key authentication. When this option is selected SiteScope uses the private key in the file **SiteScope/groups/identity** to authenticate. The corresponding public key must be listed in the **authorized_keys** file on the remote host.

➤ **Key File for SSH connections.** Select the file that contains the private key for this connection. The default key file is **SiteScope\groups\identity**. This setting only applies when the authentication method is **Key File** .

➤ **SSH Version 2 Only.** Check this option to force SiteScope to use SSH protocol version 2 only. This option only applies when using the integrated Java Client in SiteScope.

➤ **Custom Commandline.** Enter a custom commandline for a remote using the External Client. This option can be used when needing to pass specific options to the external client being executed. Valid substitution variable are:

➤ $root$: This will be translated to the SiteScope directory.

➤ $user$: This will be translated to the username entered into the remote.

➤ $password$: This will be translated to the password entered into the remote.

➤ $host$: This will be translated to the hostname entered into the remote.

➤ **SSH Port Number.** Enter the port that the remote SSH server is listening on. By default, the port number is **22**.

Understanding the Configuring Unix Remote Machine Dialog Box

You configure the Unix remote machine from the Configuring Unix Remote Machine dialog box.

➤ **Server Address.** Displays the IP address or host name of the server that you entered in the Add Machine dialog box.

➤ **OS.** Select the operating system running on the remote server. The following versions of UNIX are supported:

AIX	OPENSERVER
FreeBSD	SCO
HP/UX	SGI Irix
HP/UX 64-bit	Sun Solaris
Linux	Tru64 5.x
MacOSX	Tru64 Pre 4.x (Digital)

➤ **Connection Method.** Select the method for connecting to the server. The supported methods are:

➤ **Telnet.** Log in to the remote server using Telnet.

➤ **SSH.** Log in to the remote server using the SSH protocol. This may require additional software and setup depending on the version of UNIX you are working with.

➤ **Rlogin.** Log in to the remote server using the Rlogin protocol.

➤ **HTTP.** Connect to an HTTP server on the remote server and run the command via a CGI. For this method, the Login and Password are optional and are used for authorizing the log on to the remote machine if required.

➤ **Login.** The login for the remote server.

➤ **Password.** Enter the password for the remote server.

➤ **Title.** Enter a name by which the remote machine should be known. This name will appear in the drop-down list in monitors that can connect to this server.

➤ **Prompt.** Enter the prompt to be displayed when the system is ready to handle a command - the default is #.

➤ **Login Prompt.** Enter the prompt to be displayed when the system is waiting for the login to be entered - the default is "ogin:"

➤ **Password Prompt.** Enter the prompt to be displayed when the system is waiting for the password to be entered - the default is "assword:"

➤ **Secondary Prompt.** Enter the prompt to be displayed if the telnet connection to the remote server causes the remote server to prompt for more information about the connection. Separate multiple prompt strings by commas (,). For example, for Telnet connections to some remote servers, the remote server may ask what terminal type should be emulated for the connection. In this case you might need to enter Terminal type? as the secondary prompt. The response to the secondary prompt is entered in the Secondary Response field below.

➤ **Secondary Response.** Enter the responses to secondary prompts required to establish connections with this remote server. Separate multiple responses with commas (,).

➤ **Initialize Shell Environment.** Enter any shell commands to be executed at the beginning of the session. Separate multiple commands with a semicolon (;). This option allows you to specify shell commands to be executed on the remote machine directly after a Telnet or SSH session has been initiated. These commands can be used to customize the shell for each SiteScope remote.

Examples:

➤ The remote shell may not have the correct path set for SiteScope scripts to run. The following command will add the directory **/usr/local/bin** into the PATH of the current shell on the remote machine:

export PATH=$PATH:/usr/local/sbin

➤ The remote shell may not be initializing the pseudo terminal correctly. Enter the following command to increase the terminal width to 1024 characters:

stty cols 1024;${SHELL}

Note: Commands after a shell invocation will not be executed.

407

➤ There have been cases where the remote Telnet Server does not echo back the command line properly. This may cause strange behavior for monitors that rely on this behavior.

Enter the following command to force the remote terminal to echo:

stty echo

➤ Certain UNIX shells have been known to behave erratically with SiteScope. This includes bash, ksh, and csh. Enter the following command to change the shell to sh for the SiteScope connection:

/bin/sh

➤ **Trace.** Select this option to trace messages to and from the remote server in the **RunMonitor.log** file.

➤ **SSH Connection Method.** Select the method to use to connect to the remote server.

➤ **Internal Java Libraries.** Connect using the Java SSH client integrated with SiteScope

➤ **Plink.** Connect using an external SSH client. On Windows NT, SiteScope ships with Plink.

➤ **Disable Connection Caching.** Select this to disable SSH connection caching.

➤ **Connection Limit.** Enter the maximum number of connections for this remote machine.

➤ **SSH Authentication Method.** Select the method to use to authenticate to the remote server (for SSH connections only).

➤ **Password.** Authenticate using a password.

➤ **Keyfile.** Authenticate using public/private key authentication. When this option is selected, SiteScope uses the private key in the file **SiteScope/groups/identity** to authenticate. The corresponding public key must be listed in the **authorized_keys** file on the remote host.

➤ **Connection Limit.** Enter the maximum number of connections for this remote machine.

➤ **SSH Version 2 Only.** Select this option to force SSH to only use SSH protocol version 2. This option is only supported when using the internal Java libraries connection method.

➤ **Custom Commandline.** Enter the command for execution of the external SSH client. For substitutions with options listed above, use $host$, $user$, and $password$ respectively. This setting is supported only for connections using an external process.

➤ **SSH Port Number.** Enter the port on which the SSH service is running.

After defining the server for SiteScope, you can test the settings by clicking on the test link. SiteScope attempts to display the working directory of the remote machine (the "pwd" command on UNIX, or "cd" on Windows NT), as a test to ensure that the remote machine can be accessed and can run commands properly.

Starting the Monitors in the Controller

To start the online monitors:

 1 Start the scenario. Select the Vuser groups you want to run and click **Start Scenario**, or select **Scenario** > **Start**.

 2 Select the **Run** tab. The default graphs are displayed below the Scenario Groups window.

 3 Double-click a graph to maximize it. Repeat the operation to restore the tiled view.

4 If the graph tree is not displayed, select **View** > **Show Available Graphs**. Click the "+" in the left pane to expand the graph tree. To hide the graph tree view, select **View** > **Hide Available Graphs**.

5 Select a graph from the tree and drag it into the right pane. You can also drag graphs between panes.

Note: The Transaction Monitor graphs will not contain any data unless transactions are being executed. In addition, the other graphs will not contain any data unless you set up a list of resources to monitor before running your scenario.

Opening Online Monitor Graphs in the Controller

By default, LoadRunner displays four graphs in the Run view: Running Vusers, Transaction Response Time, Hits per Second, and Windows Resources. You can display the other graphs by clicking and dragging them from the graph tree view to the graph view area. Alternatively, you can open a new graph using the Open a New Graph dialog box.

To open a new graph using the Open a New Graph dialog box:

1 Select **Monitors** > **Online Graphs** > **Open a New Graph**, or right-click a graph and select **Open a New Graph**. The Open a New Graph dialog box opens.
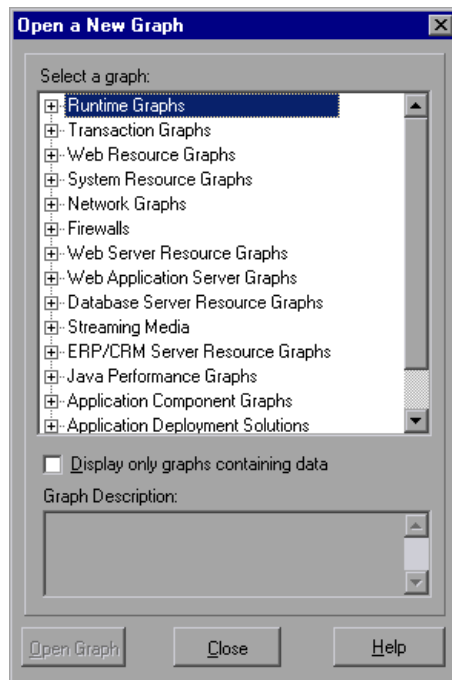
2 Click the "+" in the left pane to expand the graph tree, and select a graph. You can view a description of the graph in the **Graph Description** box.

3 Click **Open Graph**, or drag the selected graph into the right pane of the Run view. The graph appears in the graph view area.

Understanding the Open a New Graph Dialog Box

The Open a New Graph dialog box enables you to open a new graph and view its description.

➤ **Select a graph.** Click the "+" to the left of each category to expand the tree view. Select a graph.

Note: You can open only one graph at a time.

➤ **Display only graphs containing data.** Select this option to view graphs that contain data only. To view the entire list of Analysis graphs, clear this option.

➤ **Graph description.** Displays the selected graph's description.

➤ **Open Graph.** Opens the selected graph and displays it in the graph tree view.

Customizing the Online Monitor Display View

LoadRunner lets you display up to 16 online monitor graphs simultaneously.

To customize your online monitor display:

1 Right-click a graph and select **View Graphs**, or select **View** > **View Graphs**.

2 Select the number of graphs you want to view. You can select from **Show One Graph**, **Show Two Graphs**, **Show Four Graphs**, **Show Eight Graphs**, or **Custom Number**.

If you select **Custom Number**, enter the number of graphs you want to view in the View Graphs dialog box, and click **OK**. The selected number of graphs open in the graph viewing area.

To display only one graph, double-click the graph pane. To return to the previous view, double-click the graph again.

Setting Monitor Options

Before running your scenario, LoadRunner lets you configure the settings for your online monitors. You can set the data sampling rate, error handling, debugging, and frequency settings for the online monitors.

When you save a scenario, the online monitor configuration settings are saved as well.

To set monitor options:

 1 Select **Tools** > **Options** and select the **Monitors** tab.
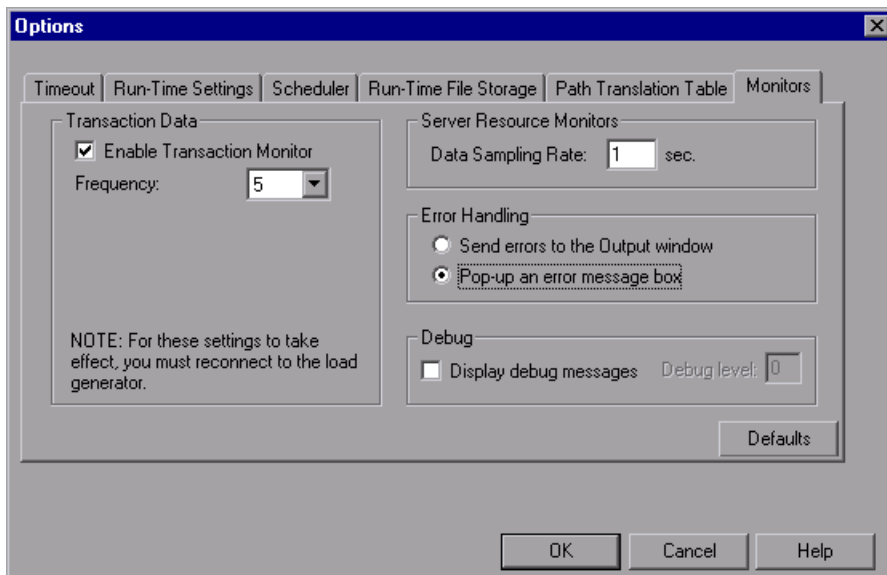
2 Ensure that **Enable Transaction Monitor** is selected (default setting), and specify the frequency at which the monitor should send updates to the Controller for the Transaction, Data Point, and Web Resource graphs.

To conserve resources, you can disable the Transaction monitor by clearing the **Enable Transaction Monitor** check box.

Note: You cannot modify these settings during scenario execution; you must stop the scenario before disabling the monitor or changing its frequency.

---

3 Enter a sampling rate.

4 Set the desired **Error Handling** option.

5 To display debug messages in the Output window, select the **Display debug messages** check box. For the Network monitor, specify a **Debug level** from 1-9.

6 Click **OK** to save your settings and close the Options dialog box.

You can configure an additional monitor setting while working in Expert mode. For information on working in Expert mode, see Appendix C, "Working in Expert Mode."

Understanding the Options - Monitors Tab

The Monitors Tab lets you enable the Transaction monitor, configure the behavior of the transaction data, and set the data sampling rate, error handling, debugging, and frequency settings for the online monitors.

➤ **Transaction Data.** Configures the behavior of data for the Transaction, Data Point, and Web Resource online graphs.

➤ **Enable Transaction Monitor.** Enables the online Vuser Transaction monitor to begin monitoring transactions at the start of a scenario.

➤ **Frequency.** Select the frequency, in seconds, at which the online monitor samples the data to produce the Transaction, Data Point, and Web Resource online graphs. The default is 5 seconds. For a small scenario, it is recommended that you use a frequency of 1. For a large scenario, it is recommended that you use a frequency of 3-5. The higher the frequency, the less network traffic there will be. The data is averaged for the frequency period defined, and only one value is sent to the Controller.

For information on enabling and disabling the Transaction monitor and Web Page Diagnostics, see the chapter "Run-Time and Transaction Monitoring" in the *HP LoadRunner Monitor Reference*.

➤ **Server Resource Monitors.** Configures the behavior of the Server Resource monitors.

➤ **Data Sampling Rate.** The sampling rate is the period of time (in seconds) between consecutive samples. Enter the rate at which LoadRunner samples the scenario for monitoring data. By default, the online monitor samples the data at intervals of three seconds. If you increase the sampling rate, the data is monitored less frequently. This setting applies to all graphs. To set a sampling rate for a specific graph, see "Configuring Graph Properties" on page 421.

Note: The data sampling rate you set is applied to all server monitors that you subsequently activate. It is not applied to server monitors that have already been activated. To apply the new data sampling rate to activated server monitors, save your scenario and reopen it.

Each monitor has a different minimum sampling rate. If the default sampling rate, or the rate set in the Options Monitors tab is less than a monitor's minimum sampling rate, the monitor will sample data at intervals of its minimum sampling rate. For example, the minimum sampling rate for the Oracle Monitor is 10 seconds. If the sampling rate in the Options Monitors tab is set at less than 10 seconds, the Oracle Monitor will continue to monitor data at 10 second intervals.

➤ **Error Handling.** Controls the way in which LoadRunner issues error messages. Select one of the following options:

  ➤ **Send errors to the Output window.** Sends all errors to the Output window.

  ➤ **Pop-up an error message box.** Sends errors to a message box (default). To dismiss the message box, click **OK**.

➤ **Debug.** For debugging a scenario, you can set the following option:

  ➤ **Display debug messages.** Sends debug-related messages to the output log. You can also specify a debug level from 1-9. The debug level is only relevant to the Network monitor.

25

Configuring Online Graphs

You can view the data collected by the monitors using the LoadRunner online monitor graphs.

This chapter includes:

➤ About Online Monitor Graphs on page 420

➤ Configuring Graph Properties on page 421

➤ Configuring Graph Measurements on page 425

➤ Merging Graphs on page 431

➤ Exporting Online Monitor Graphs on page 432

➤ Viewing Data Offline on page 433

➤ Available Graphs Tree on page 433

About Online Monitor Graphs

Online monitor graphs display performance measurements for those resources being monitored during scenario execution. Each measurement is represented on the graph by a colored line. Information about the measurements is listed in the legend below the graph. The legend displays the measurements for the selected graph only.

For more information on opening monitor graphs and customizing the display, see "Opening Online Monitor Graphs in the Controller" on page 412, and "Setting Monitor Options" on page 414.
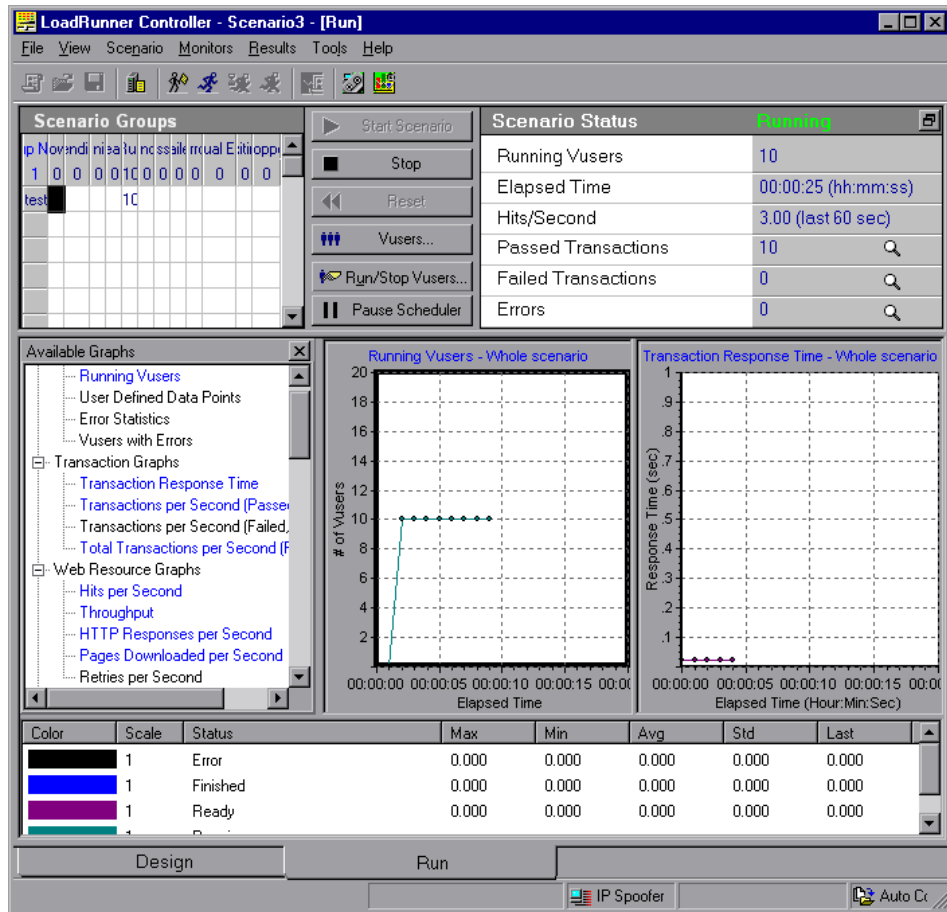
To get additional information about a measurement, right-click the measurement and select **Description**.

To focus on a particular line, you can:

➤ **Highlight a measurement.** To highlight a specific measurement, select it in the legend. The corresponding line in the graph is displayed in bold.

➤ **Hide a measurement.** To hide a measurement, right-click the measurement and select **Hide**. To hide all measurements other than the measurement selected, right-click the measurement and select **Show Only Selected**. To show a hidden measurement, right-click the measurement and select **Show**.

➤ **Pause the monitor.** To pause a specific graph during scenario execution, select the graph and select **Monitors** > **Online Graph** > **Freeze,** or right-click the graph and select **Freeze**. To resume, repeat the above action. When you resume, the graph displays the data for the paused period.

To maintain the sort order after the legend is refreshed, right-click the graph and select **Keep Legend Sorted**. Click again to remove the sort order. The legend is refreshed every five seconds.

Configuring Graph Properties

LoadRunner lets you configure the settings for your online monitor graphs. You can customize your graph in the following areas:

➤ Refresh Rate

➤ Time

➤ Graph Time

➤ Display Type

➤ Bar Values Type

➤ Y-Axis Scale

➤ Network Delay View

Note: These settings can be set globally—to apply to all graphs—or per graph.

To customize your graphs:

1 Select the online graph you want to configure (in either the right or left pane) and select **Monitors** > **Online Graphs** > **Configure**. Alternatively, right-click a graph and select **Configure**. The Graph Configuration dialog box opens.
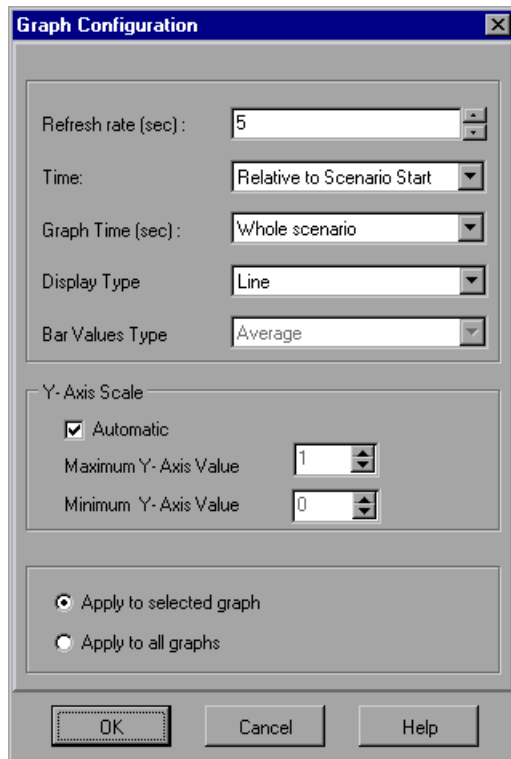
2 Enter the desired refresh rate—the time between graph updates—in the **Refresh Rate** box.

3 Select a style for the x-axis from the **Time** box.

4 Select a value from the **Graph Time** box. The graph time is the time in seconds displayed by the x-axis.

5 Select a graph style from the **Display Type** box.

6 If the selected display type is **Bar**, select a value from the **Bar Values Type** box. This determines the type of value that will be displayed in the bar graph. You can select between **Average**, **Last Value**, **Minimum** and **Maximum**.

7 Select a maximum or minimum value for the y-axis, or select **Automatic** to view graphs using the default y-axis scale.

8 To apply the dialog box settings to all graphs, select **Apply to all graphs**.

9 For the Network Delay Time graph, you can select the following options:

➤ **SubPaths.** Displays the delay measurements from the source machine to each of the nodes along the network path.

➤ **DNS name.** Displays the DNS names of the measurements in the legend.

10 Click **OK** to save your settings and close the Graph Configuration dialog box.

Understanding the Graph Configuration Dialog Box

The Graph Configuration dialog box lets you customize the online graph settings.

➤ **Refresh Rate.** The interval at which the graph is refreshed with new data. By default, the graph is refreshed every five seconds. If you increase the refresh rate, the data is refreshed less frequently.

Note: In a large load test, it is recommended to use a refresh rate of three to five seconds. This enables you to avoid problems with CPU resource usage.

Time: You can specify how the graph displays the x-axis time.

➤ **Don't Show.** Instructs LoadRunner not to display values for the x-axis.

➤ **Clock Time.** Displays the absolute time, based on the system clock.

➤ **Relative to Scenario Start.** Displays the time relative to the beginning of the scenario.

Note: If no step is running, clock time is displayed.

In the following example, the graph is shown with the **Don't Show** and **Clock Time** options:
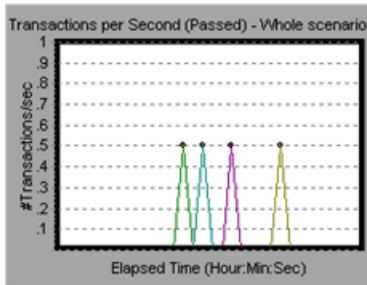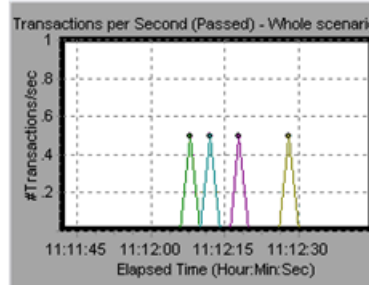
Don't Show   *Clock Time*

➤ **Graph Time.** Indicate the scale for a graph's x-axis when it is time-based. A graph can show 60 to 3600 seconds of activity. To see the graph in greater detail, decrease the graph time. To view the performance over a longer period of time, increase the graph time. The available graph times are: **Whole Scenario**, **60**, **180**, **600**, and **3600** seconds.

➤ **Display Type.** Specify whether LoadRunner displays a graph as a line graph or a bar graph. By default, each graph is displayed as a line graph.

Note: For the Network Delay graph, if you select **View Segments**, you can view the network segments of the graph as an area graph or a pie graph.

➤ **Bar Values Type.** Select a value from the Bar Values Type box (if the selected display type is Bar). This determines the type of value that will be displayed in the bar graph. You can select between **Average**, **Last Value**, **Minimum** and **Maximum**.

➤ **Y-Axis Scale.** Instruct LoadRunner to display graphs using the default y-axis scale, or you specify a different y-axis scale. Click **Automatic** if you want LoadRunner to use the default y-axis values. Specify a maximum or minimum value for the y-axis if you want to modify the y-axis scale.

➤ **Network Delay View.** This option only appears when you configure the Network Delay Time graph. Click **SubPaths** to view the delay measurements from the source machine to each of the nodes along the network path. Click **DNS name** to view the DNS names of the measurements displayed in the legend.

Configuring Graph Measurements

You can configure the following online measurement settings:

➤ Change Line Colors

➤ Set Measurement Scale

➤ Show and Hide Transactions

Changing Line Colors

LoadRunner assigns a unique color to each measurement. You can modify the color using the configuration interface.

To change the line color of a measurement:

 1 In the legend below the graphs, select the measurement you want to configure. Right-click and select **Configure**. The Measurement Configuration dialog box opens.
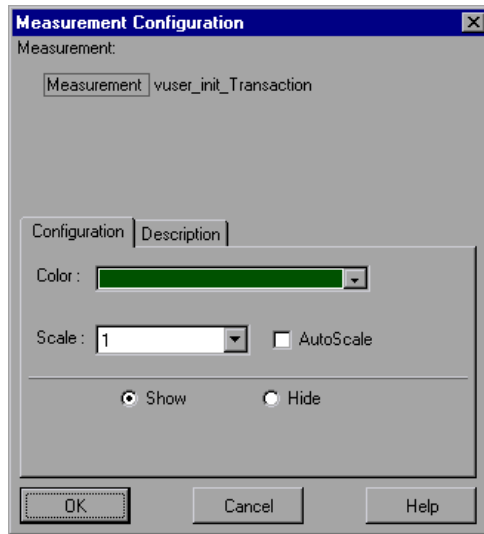
 2 To change the color of the line, select a color from the **Color** list.

 3 Click **OK** to accept the settings and close the dialog box.

The specified color changes are reflected in the graph and in the legend beneath the graph. The color is displayed in the first column of the legend.

Color	Scale	Transaction	Max	Min	Avg	Std	
	10	DOGBER...	0.5	0	0.013158	0.080036	
Hidden	10	Mercury_I...	0.5	0	0.013158	0.080036	
	10	Mercury_I...	0.5	0	0.013158	0.080036	
	10	Mercury I...	0.5	0	0.045455	0.14374	

Setting the Scale of the Measurement

You can modify the scale of a measurement—the relationship between the y-axis and the graph's actual value. For example, a scale set at 1 indicates that the measurement's value is the value of the y-axis. If you select a scale of 10, you must divide the y-axis value by 10 to obtain the true value of the measurement.

To set the scale of a measurement:

1 Select the measurement you want to configure. Right-click and select **Configure**. The Measurement Configuration dialog box opens.

2 Clear the **Autoscale** check box and select the desired ratio from the Scale list.

3 Click **OK** to accept the settings and close the dialog box.

In the following example, the same graph is displayed with a scale of 1 and 10:

Scale =1   Scale = 10
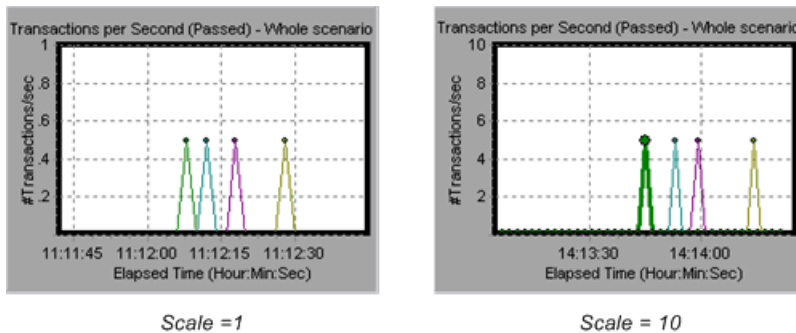
The actual graph values range from 0-1, as shown in the left graph. You can view the information more accurately using a larger scale for the display, as shown in the right graph. However, to obtain the actual values, you need to divide the displayed value by the scale. In the example above, the highest value shown in the graph is 5. Since the scale is 10, the actual value is 0.5.

The legend below the graph indicates the scale factor.

By default, LoadRunner uses the **autoscale** option, which automatically scales the measurements by calculating the best ratio for displaying the graph.

Hiding and Showing Transactions

By default, the Transaction Monitor displays a line for each item in the transaction list. You can hide the line for any of the monitored transactions to focus on a specific transaction.

To hide or show a transaction:

1 To hide a transaction, click **Hide**. To show a hidden resource, click **Show**. To hide all transactions other than the transaction selected, right-click the transaction and select **Show Only Selected**.

2 Click **OK** to accept the settings and close the dialog box.

You can also show and hide transactions without opening the Measurement Configuration dialog box, by right-clicking a transaction in the legend and selecting **Show/Hide**.

In the following example, a line is shown for each transaction:
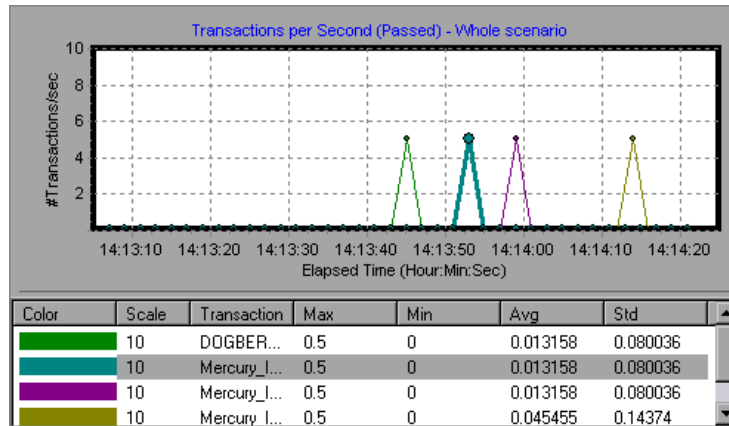
In this example, the second item in the legend is hidden:
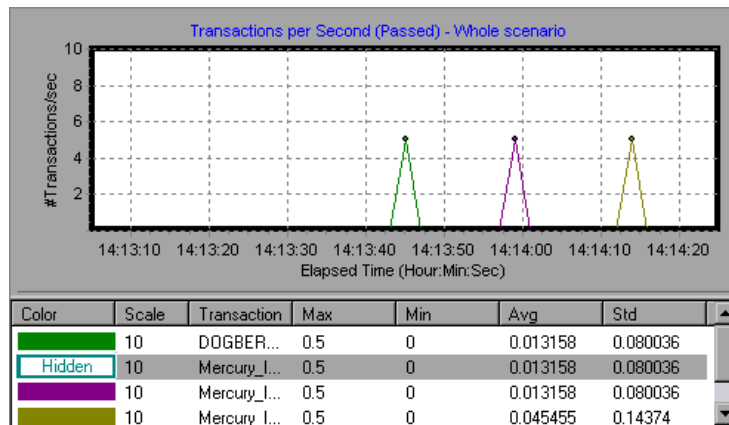
Understanding the Measurement Configuration - Configuration Tab

The Measurement Configuration tab lets you change line colors, set the scale of a measurement, and show or hide transactions.

➤ **Measurement.** Displays the type of resource being monitored.

➤ **Machine.** Displays the name of the machine whose resources are being monitored (appears only in cases where a machine's resources are being monitored).

Note: When monitoring a network path, the **Network Type** will appear here instead of **Machine**.

➤ **Color.** Select a color to be assigned to the selected measurement.

➤ **Scale.** Displays the relationship between the y-axis and the graph's actual value. For example, a scale set at 1 indicates that the measurement's value is the value of the y-axis. If you select a scale of 10, you must multiply the y-axis value by 10 to obtain the true value of the measurement.

➤ **Autoscale.** Instructs LoadRunner to automatically scale the measurement by calculating the best ratio for displaying the graph. For some graphs, this option is not available.

➤ **Show.** Shows the selected resource. The line for the selected resource reappears in the graph. By default, all resource measurements are displayed in the chart.

➤ **Hide.** Hides the selected resource. The line for the selected resource disappears from the graph. The hidden resources are displayed as unfilled boxes.

Understanding the Measurement Configuration - Description Tab

The Measurement Description tab displays information about the measurement.

➤ **Measurement.** Displays the type of resource being monitored.

➤ **Machine.** Displays the name of the machine whose resources are being monitored (appears only in cases where a machine's resources are being monitored).

➤ **Description.** Displays a description of the selected measurement.

Merging Graphs

LoadRunner lets you merge the results of two graphs from the same scenario into a single graph. The merging allows you to compare several different measurements at once. For example, you can make a merged graph to display the Web Throughput and Hits per Second, as a function of the elapsed time. In order to merge graphs, the x-axis of both must be the same measurement.

When you overlay the contents of two graphs that share a common x-axis, the left y-axis on the merged graph shows the current graph's values. The right y-axis shows the values of the graph that was merged.

To overlay two graphs:

1 Right-click one of the graphs you want to overlay, and select **Overlay Graphs**. The Overlay Graphs dialog box opens.
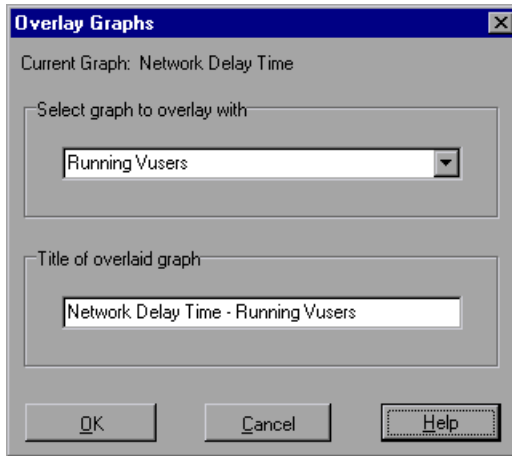
2 Select a graph with which you want to overlay the current graph. The drop-down list shows only the active graphs that have a common x-axis with the current graph.

3 Enter a title for the overlaid graph.

4 Click **OK**. The merged graph appears in the graph view area.

Exporting Online Monitor Graphs

LoadRunner allows you to export online graphs to HTML for viewing at a later stage. When you export to HTML, the legend is also displayed with the graph.

To export online graphs to HTML:

1 To export all graphs in the Online Monitor view, select **Monitors** > **Export Online Graphs to HTML.** The Select Filename and Path dialog box opens.

2 Specify a filename and path and click **Save**.

Viewing Data Offline

After monitoring resources during a scenario run, you can view a graph of the data that was gathered using Analysis. When you run the Analysis utility, it processes the data and generates a graph for each measurement that was monitored.

To view a graph, select **Graph > Add Graph** in the Analysis window. For more information about working with Analysis at the conclusion of the scenario, see the *HP LoadRunner Analysis User's Guide*.

Available Graphs Tree

The Available Graphs Tree displays the LoadRunner graphs.

To open a graph, click the graph in the graph tree, and drag it into the right pane of the Run view.

To select the measurements that you want to monitor on the graph, see the monitor configuration instructions for the specified monitor in the *HP LoadRunner Monitor Reference*.

Part VII

Appendixes

A

Interpreting LoadRunner Online Graphs

LoadRunner online monitor graphs present important information about the performance of your scenario. This appendix describes some of the key online graphs in greater depth, and shows how they can be used to identify and pinpoint performance bottlenecks as your scenario is running.

This chapter includes:

➤ Online Monitor Graphs on page 438

Online Monitor Graphs

Using the online monitor graphs, you can determine whether transactions transpire within an acceptable amount of time, whether your bandwidth is sufficient to keep download times to a minimum, and whether your hardware and operating system can handle peak load.

Question 1: Do all transactions in my scenario transpire within an acceptable amount of time? Which particular transactions take too long?

Answer: The **Transaction Response Time** graph shows the amount of time it takes for each transaction to be completed. In the graph below, the transaction response time is quick, except for the login transaction. The initial login did not take much time, but subsequent logins were quite slow. This indicates that the database is unable to process more than one login at a time, which may be due to inefficient database querying.

Color	Scale	Transaction	Max	Min	Avg	Std
	1	Mercury_Tours	9.944	1.332	4.2088	2.3487
	1	Welcome_to	1.713	0.49	0.9224	0.35404
	1	Find_Flights	3.114	0.331	1.6353	1.0861
	1	Search_Results	1.743	0.25	1.0477	0.51256
	1	Method_of_Payment	1.953	0.32	1.1756	0.50171
	1	Flight_Confirmation	2.253	0.231	1.1041	0.63992
	1	Mercury_Tours_2	2.013	0.32	0.77321	0.5076
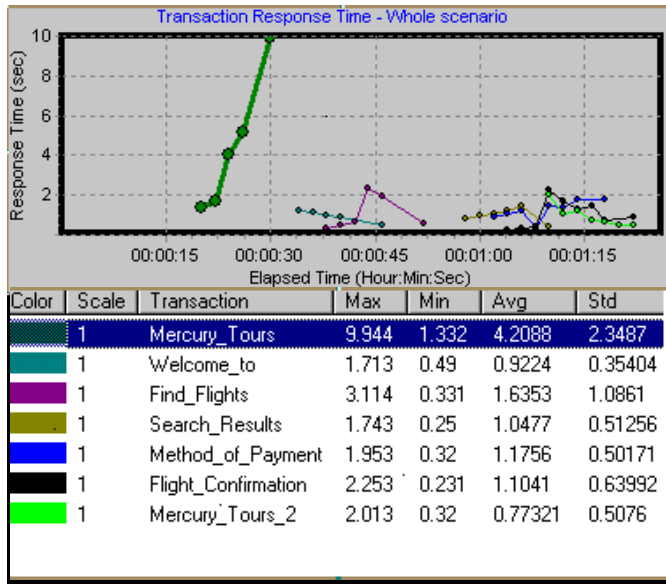
Question 2: Is bandwidth sufficient to keep download times to a minimum?

Answer: The **Throughput** graph shows the amount of throughput on the Web server during each second of the scenario run. Throughput represents the amount of data received from the server at any given second.
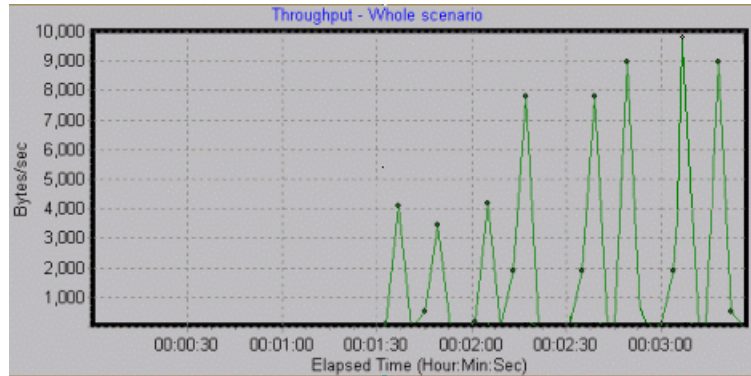
In the above graph, the throughput scales upward as time progresses and the number of users increases, indicating that the bandwidth is sufficient. If the graph were to remain relatively flat as the number of users increased, it would be reasonable to conclude that the bandwidth is constraining the volume of data requested.

Question 3: Can the hardware and operating system handle peak load?

Answer: The **Windows Resources** graph displays Windows resource usage in real time. You use this graph to monitor the resources used during a scenario and locate a bottleneck on a particular machine.
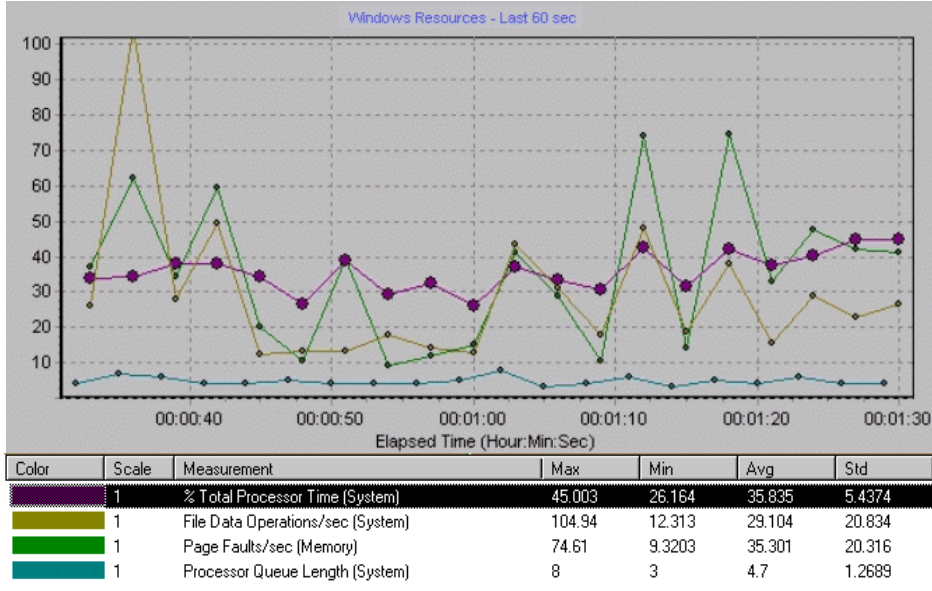
The **% Total Processor Time** in the above graph shows the amount of data processed by the server. **File Data Operations/sec** shows the rate at which the server is issuing Read and Write operations to file system devices. **Page Faults/sec** counts the number of page faults in the processor, representing virtual memory and caching algorithm opportunities.

It is commonly thought that newer and faster servers can resolve slow download times. However, the above graph demonstrates that only a small amount of data is processed by the server. The graph indicates that there is adequate processor capacity, and additional server hardware will not result in increased performance. There are cases, however, in which increased performance can be achieved by optimizing the data file system.

B

Performing Path Translation

When you run a scenario, LoadRunner gathers run-time data from the participating Vusers. By default, LoadRunner stores the data in temporary files on each Vuser machine. After the scenario, the data is collated in the general results directory.

Alternatively, you can instruct LoadRunner to write the run-time data directly to a shared network drive. (See Chapter 11, "Configuring a Scenario.") This method is not recommended, since it increases network traffic and necessitates path translation.

This chapter includes:

➤ Understanding Path Translation on page 444

➤ Adding Entries to the Path Translation Table on page 445

➤ Editing the Path Translation Table on page 447

➤ Path Translation Examples on page 448

Understanding Path Translation

Path Translation is a mechanism used by LoadRunner to convert a remote path name for the Controller. A typical scenario might have the Controller running on a Windows-based machine and include multiple Vusers running on both Windows-based and UNIX load generators. One remote load generator may map the network drive as F, while another load generator maps the same drive as H. In a complex scenario such as this, you need to ensure that all participating machines recognize the same network drive.

You instruct LoadRunner to store scripts and run-time data results on a shared network drive from the Run-Time File Storage tab of the Options dialog box.
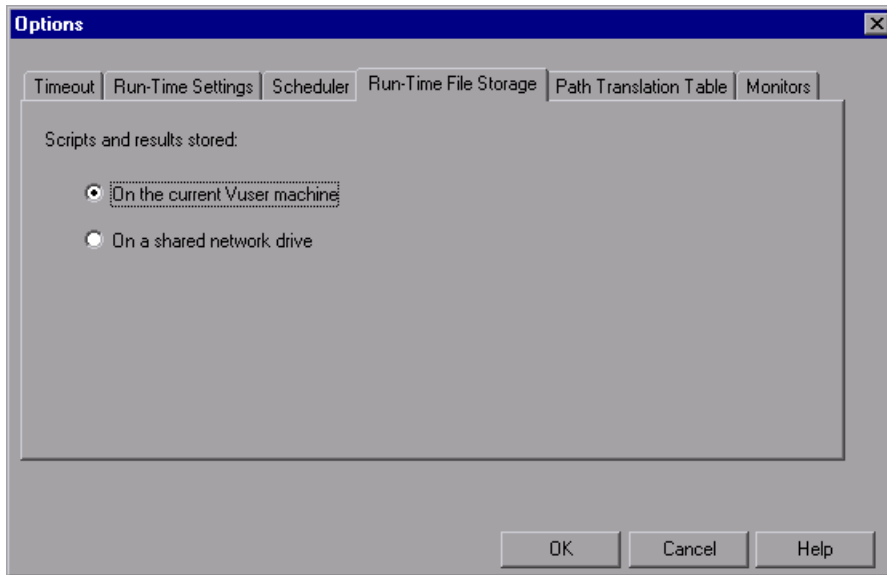
Result and script files stored on a shared network drive require you to perform path translation.

The Scenario Groups/Scripts pane in the Design view contains a list of all the Vuser scripts associated with a scenario—and their locations. A script's location (path) is always based on the Controller machine's mapping of that location. If a Vuser load generator maps to the script's path using a different name, path translation is required.

For example, assume that the Controller is running on a Windows-based machine named pc2, and that a Vuser script is located on a network drive. The Controller machine maps the network drive as **m:\lr_tests**. If the remote Vuser machine (load generator) hosting the Vusers also maps the path as **m:\lr_tests**, no translation is necessary. However, if the remote machine maps the path as another drive or path, for example **r:\lr_tests**, you must translate the path to enable the load generator to recognize the script location.

Similarly, when saving run-time result files to a shared drive that is mapped differently by the Controller and remote load generator, you must perform path translation.

Path translation is also effective across platforms—between Windows and UNIX. You use path translation to translate Windows-based paths (as seen by the Controller) into paths recognized by the UNIX Vuser load generator.

Adding Entries to the Path Translation Table

To translate a path from one Windows-based computer to another, or between Windows-based and UNIX machines, you create an entry in the Path Translation table. This table contains a list of paths translated into formats that can be recognized by different machines.

Each line of the Path Translation table has the following format:

<controller_host> <controller_path> <remote_path> [<remote_host>]

➤ **controller_host.** The name or type of the machine that is running the Controller. For example, if the Controller is running on a Windows-based computer, you could type win in the host field. Alternatively, you could enter the name of the machine running the Controller (for example, LOADPC1).

The value of **controller_host** can be:

➤ **hostname.** The name of the machine running the Controller

➤ **win.** The Controller is running on a Windows-based computer

➤ **unix.** The Controller is running on a UNIX machine

➤ **all.** The Controller is running on a Windows-based or a UNIX machine

➤ **controller_path.** The path of a specific directory—as recognized by the Controller. For example, if the directory **scripts** is located on the network drive **r**—as mapped by the Controller—type the path r:\scripts in the **controller_path** field.

➤ **remote_path.** The path of a specific directory—as recognized by the remote machine. For example, if the directory **scripts** is located on the network drive **n**—as mapped by the remote load generator—type the path n:\scripts in the **remote_path** field.

If a Vuser on the remote UNIX load generator recognizes the above path as /m/tests, you would type this path in the **remote_path** field.

➤ **remote_host.** The name or type of the remote load generator. For example, if all the remote machines are UNIX workstations, you could type unix in the **remote_host** field. The options for the **remote_host** field are the same as the options for the **controller_host** field, listed above. The **remote_host** parameter is optional.

Editing the Path Translation Table

You maintain the Path Translation table using the Controller. LoadRunner saves the Path Translation table as an ASCII file, **ppath.mnt.** This file, stored in **_directory/dat**, has a one–line entry for each network path to translate.

To edit the Path Translation table:

 1 Start the Controller.

 2 Select **Tools** > **Options** and select the **Path Translation Table** tab. The Path Translation Table view opens.
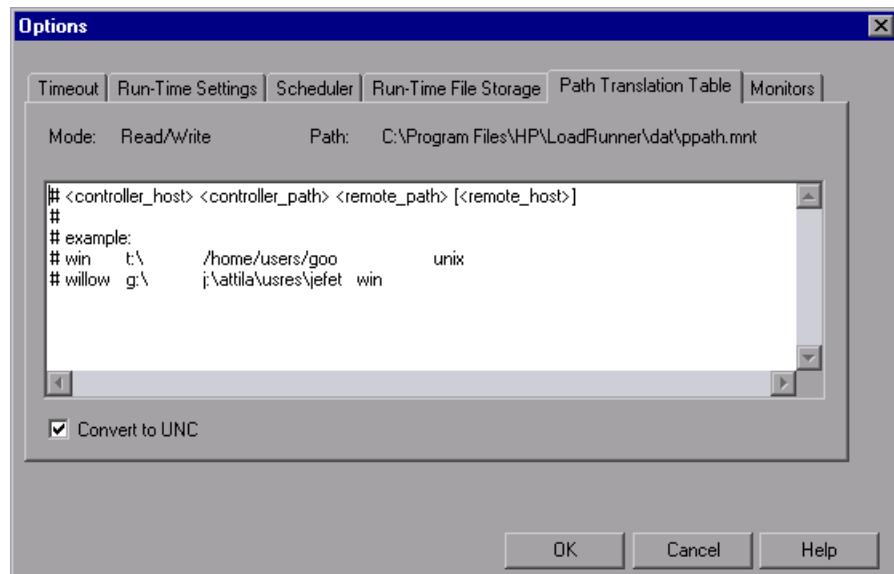
 3 Before you enter path translation information, consider using the Universal Naming Convention method. If your machines are Windows machines, you can tell the Controller to convert all paths to UNC, and all machines will be able to recognize the path without requiring path translation. An example of UNC format is \\machine_a\results.

Select the Convert to UNC check box to tell LoadRunner to ignore the path translation table and to convert all paths to the Universal Naming Convention.

4 If your machines are not Windows machines and you require path translation, type the path information into the table. You can insert comments by typing the "**#**" symbol at the start of a line in the table.

5 Click **OK** to close the table and save the information.

Path Translation Examples

The following section illustrates sample Path Translation Table entries.

Note: When you translate a Windows-based path to a UNIX path, you must enter the appropriate slashes—forward slashes for UNIX and back slashes for Windows-based paths.

The examples below show the use of the Path Translation table for a Windows-based Controller called Merlin.

In the first example, Vusers are running on a Windows 2000 machine, Oasis. Merlin maps the network drive as f:, while Oasis maps it as g:\loadtest.

merlin	f:\	g:\loadtest\	Oasis

In the second example, Vusers are running on a UNIX machine, Ultra. Ultra maps the networks drive as /u/tests/load.

merlin	f:\	/u/tests/load/	Ultra

In the third example, the mapping of the network drive by the remote load generator Jaguar, is identical to the Controller's mapping, so no translation is required. This line can be excluded from the Path Translation table.

merlin	n:\	n:\	Jaguar

In the fourth example, all Windows-based Vuser load generators map the
network drive as m:\loadtest.

merlin	l:\mnt\	m:\loadtest\	win

C

Working in Expert Mode

Advanced users can fine-tune the LoadRunner configuration settings while working in **Expert Mode**. In Expert mode, additional options are displayed in the Options dialog box.

This chapter includes:

➤ Entering Expert Mode on page 451

➤ Options - General Settings on page 452

➤ Options - Debug Information Settings on page 454

➤ Options - Output Settings on page 456

➤ Options - Monitor Settings on page 458

➤ Load Generator Information - UNIX Environment Settings on page 459

➤ Load Generator Information - Connection Log Settings on page 460

Entering Expert Mode

The HP LoadRunner Controller Expert mode is intended to provide support personnel with access to system information. When you work in the Expert mode, the Controller dialog boxes contain additional options for fine tuning the Controller operation.

To activate the Expert mode, select **Tools** > **Expert Mode.** An active Expert mode is indicated by a check mark.

To exit the Expert mode, repeat the above process.

Options - General Settings

The General tab in the Options dialog box allows you to specify global settings for data table storage and multiple IP address allocation, and instruct LoadRunner not to collate log files. This tab is displayed only when you operate the Controller in the Expert mode.

To set the General Expert mode settings:

1 Select **Tools** > **Options**. The Options dialog box appears. Select the **General** tab.
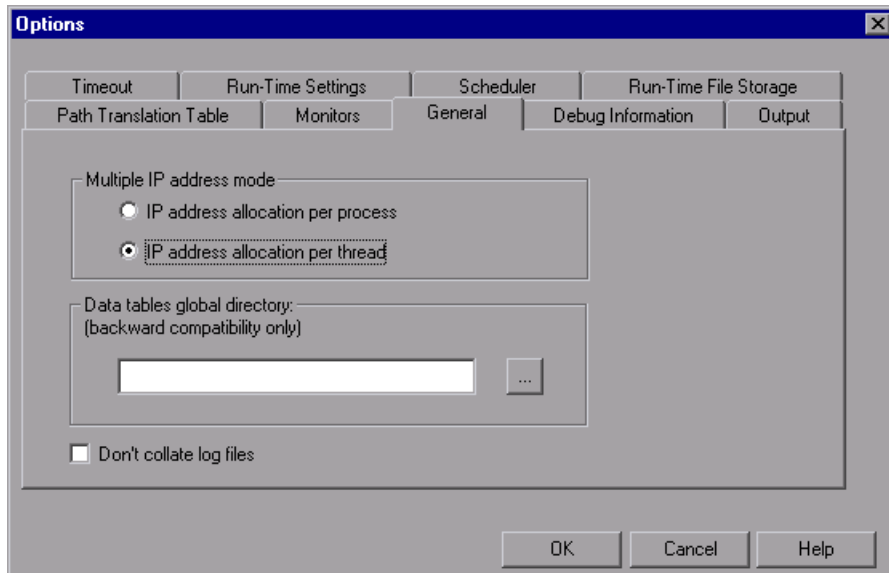
2 Select the Multiple IP address mode.

3 Enter the global directory for data tables.

4 If you want LoadRunner to collate only result files and not log files, select **Do not collate log files**.

5 Click **OK** to accept the settings and close the dialog box.

Understanding the Options - General Tab

The General tab allows you to specify global settings for data table storage, log file collation, and multiple IP address allocation.

➤ **Multiple IP address mode.** The mode used to allocate IP addresses when the multiple IP address option is enabled (**Scenario** > **Enable IP Spoofer**). The Controller can allocate an IP address per process or per thread. Allocation per thread results in a more varied range of IP addresses in a scenario.

➤ **Data tables global directory.** The network location for data tables used as a source for parameter values. This setting is only required for scripts created with earlier versions of .

➤ **Do not collate log files.** Instructs LoadRunner to collate only result files, and not log files.

Options - Debug Information Settings

The Debug settings in the Options dialog box allow you to determine the extent of the trace to be performed during scenario execution. The debug information is written to the Output window. This tab is displayed only when you operate the Controller in the Expert Mode.

To set the Debug Information settings:

1 Select **Tools** > **Options**. The Options dialog box appears. Select the **Debug Information** tab.
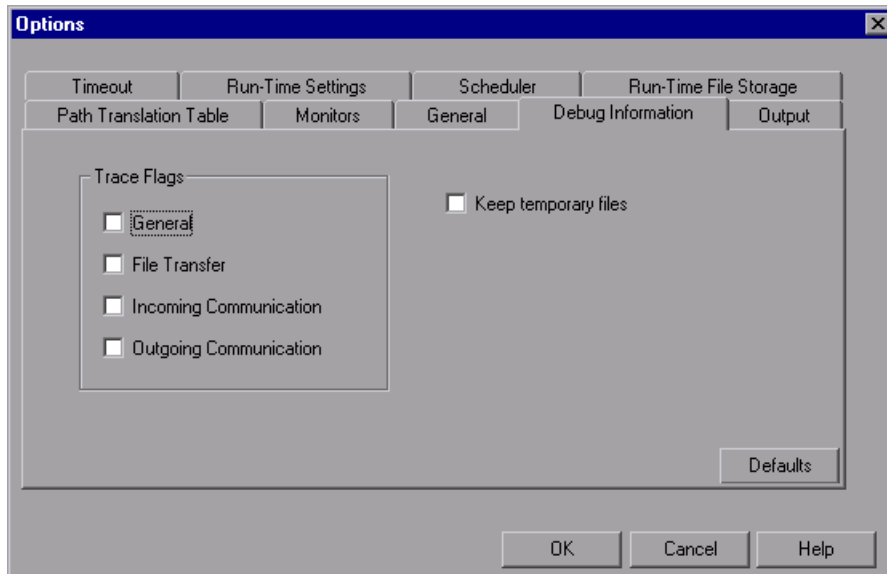
2 Select the check boxes for the desired trace flags.

3 To save the temporary run-time files, select **Keep temporary files**.

4 Click **OK** to accept the settings and close the dialog box.

Understanding the Options - Debug Information Tab

Allows you to define the LoadRunner Debug configuration. This tab is displayed only when you operate the Controller in the Expert Mode.

➤ **Trace flags.** For debugging purposes, you can configure the type of trace performed by LoadRunner during test execution. Select the appropriate check box(es) to enable the detailed trace. The trace information appears in the log file located in the specified Agent log directory. The available trace flags are: **General**, **File Transfer**, **Incoming Communication**, and **Outgoing Communication**. Select only the flags relating to your problem. For example, if you encounter specific problems with the transfer of files, select the File Transfer flag.

➤ **Keep temporary files.** The Agent and Controller create temporary files that collect information such as the parameter file sent to the Vuser, the output compilation file, and the configuration file. The Agent files are saved in **brr** folders in the TMP or TEMP directory of the Agent machine. The Controller files are saved in **lrr** folders in the TMP or TEMP directory of the Controller machine. At the end of the scenario, all these files are automatically deleted. The **Keep temporary files** setting instructs the Agent and Controller not to delete these files if you need them for debugging.

Options - Output Settings

The Output tab in the Options dialog box allows you to configure how running Vusers are displayed on the Controller machine. This tab is displayed only when you operate the Controller in the Expert Mode.

To set the Output settings:

 1 Select **Tools** > **Options**. The Options dialog box appears. Select the **Output** tab.
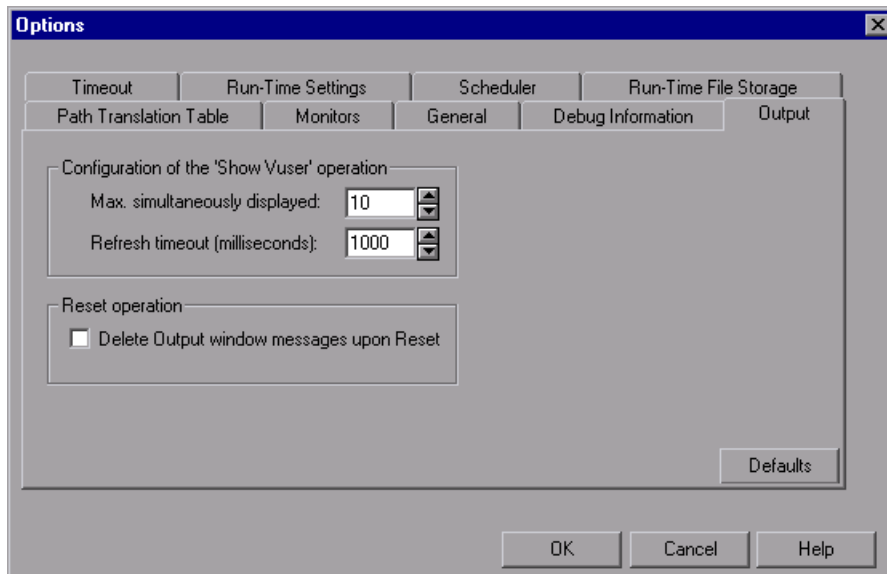
 2 In the **Max. simultaneously displayed** box, specify the maximum number of Vuser logs to be displayed simultaneously.

 3 In the **Refresh timeout** box, specify the frequency at which LoadRunner refreshes the Vuser log.

 4 To clear the messages in the Output window when you reset a scenario, select the **Delete Output window messages upon Reset** check box.

 5 Click **OK** to accept the settings and close the dialog box.

Understanding the Options - Output Tab

The Output tab enables you to configure the display of running Vusers on the Controller machine.

➤ **Configuration of the 'Show Vuser' Operation.**

➤ **Max. simultaneously displayed.** Specifies the maximum number of Vuser logs that may be displayed simultaneously, as well as the maximum number of active UNIX, GUI, RTE, or Web Vusers that the Controller should display by opening up Run-Time Viewers on your machine. The default number is ten.

➤ **Refresh timeout (milliseconds).** Defines how often to refresh the Vuser log. The default is every 1000 milliseconds.

➤ **Delete Output window messages upon Reset.** Instructs LoadRunner to clear all messages in the Output window when you reset a scenario.

Options - Monitor Settings

Expert Mode provides the following additional monitor setting:

➤ **Send Summary or Raw Data.** Sends a summary of the collected data back to the Controller, or sends all of the data in raw form. Sending the data in raw form saves time because the data does not need to be processed. However, since all of the data is being transferred to the Controller, it may cause more network traffic. If the transfer speed is significant to you, it is recommended that you select **Summary**.
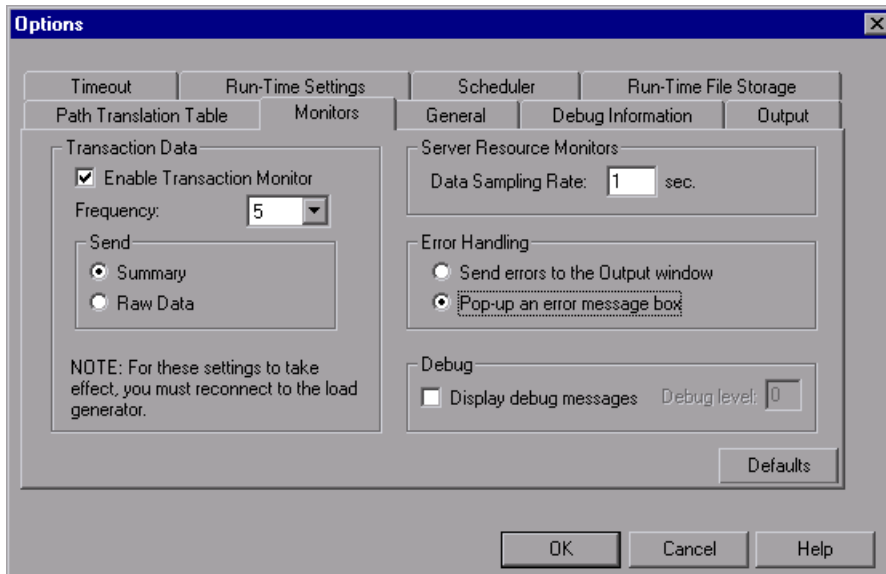
Load Generator Information - UNIX Environment Settings

Expert mode provides the following additional UNIX Environment setting:

➤ **Local User.** UNIX load generators that use the **rsh** shell establish a connection as the current NT user (due to security considerations). To "mislead" rsh and log in as a user other than the current NT login, select the **Local user** check box and specify the desired UNIX login name. Since modifying the local user name is a security breach for **rsh**, this option should be used only when you encounter a problem connecting to the remote machine.

Load Generator Information - Connection Log Settings

The Connection Log tab in the Load Generator dialog box allows you to view the standard output and standard errors generated as the Controller connects to the selected UNIX load generator. You can also change the command that the Controller sends to the remote bridge in order to connect to the load generator.
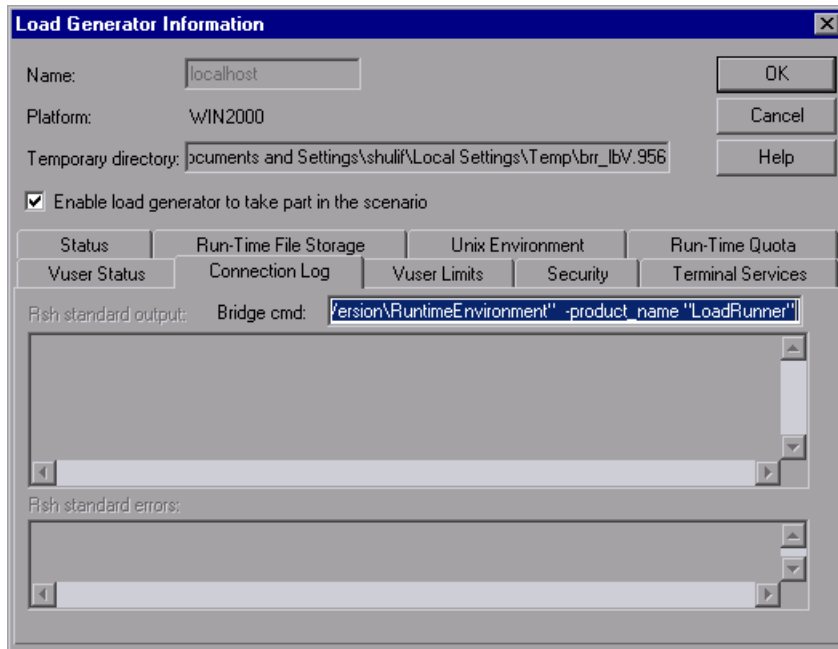
➤ **Rsh standard output.** Displays rsh standard output as the Controller connects to the selected UNIX load generator.

➤ **Bridge cmd.** Enter a new command if you want to change the default bridge command being sent by the Controller to the remote bridge in order to connect the UNIX load generator.

➤ **Rsh standard errors.** Displays rsh standard errors as the Controller connects to the selected UNIX load generator.

To set the Connection Log settings:

1 Click the **Generators** button, or select **Scenario** > **Load Generators**. The Load Generators dialog box opens.

2 Click **Connect** to change the status of a load generator from Down to Ready.

3 Click the **Details** button. The Load Generator Information dialog box opens. Select the **Connection Log** tab.

4 View the rsh standard output and rsh standard errors, or enter a new command in the Bridge cmd box to change the default bridge command.

D

Troubleshooting the Controller

LoadRunner enables you to test entire applications. If one of the components of the application is not configured properly, LoadRunner scenarios will not run.

This chapter includes:

About Troubleshooting

LoadRunner relies heavily upon communication between machines in a network. If communication is not established properly, the Controller will be unable to send commands to remote load generators and the scenario will fail. By understanding the reason for the failure and determining when the failure occurred, you can solve most network communication-related problems.

To ensure that the problem lies with your scenario and not your Vuser script, you should verify that your script runs properly on all remote load generators in stand-alone mode:

➤ Test your GUI Vuser scripts on Windows platforms using WinRunner.

➤ Test your Vuser scripts on UNIX platforms by running them from the command line.

➤ Test all other types of Vuser scripts on Windows platforms by running them from VuGen, or by running a single user from the Controller.

Note: When a test runs in VuGen, the full browser is used. This differs from a test run in the Controller, where only the browser basics are used. There may be occasions when a test passes its run in VuGen, but fails when it is run in the Controller. Before running a scenario in the Controller with multiple Vusers, run a single Vuser to ensure the test is bug free.

For more information on running Vuser scripts in stand-alone mode, see the appropriate guide for creating Vuser scripts.

LoadRunner Communications

Most communication problems can be solved if you understand your LoadRunner configuration. This knowledge helps you to determine the source of the problem and perform the necessary actions to correct it.

The following diagram illustrates a sample network running . There are five servers: The Controller, the Web server, the application server, the database server, and the file server that stores the scenario results. (Result files can also be saved on a non-dedicated server.) There are five remote load generators, each running multiple Vusers.

The arrows indicate the type of communication necessary between the elements of the network. The Vusers communicate with the Controller in both directions (send/receive), but with the file server in one direction (send). The Controller must have access to the file server. All Vusers participating in the scenario must be able to communicate with the Web server in both directions (send/receive). For a client machine to connect to the server machine, it must be able to resolve the server machine name.
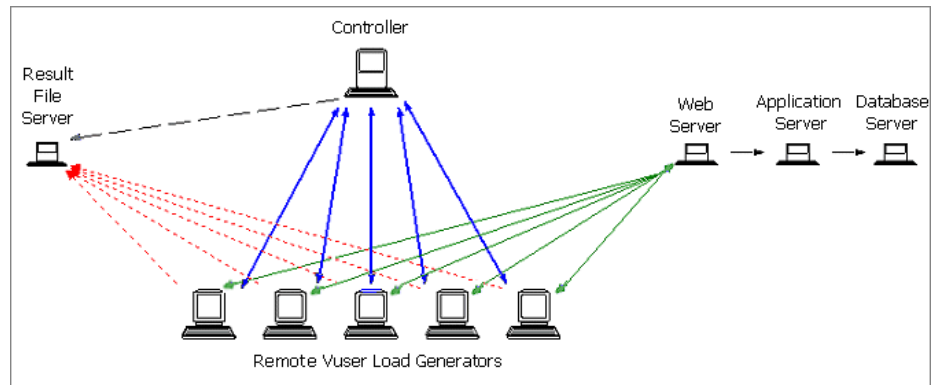
If any of the connections is broken, the scenario will fail.

Failure to Communicate with a Load Generator

The most common communication error is the failure of the Controller machine to connect with a remote load generator. Check the following items:

➤ TCP/IP connectivity

➤ Load generator connections

➤ UNIX shell

Checking TCP/IP Connectivity

Make sure that TCP/IP connectivity is functional on the Controller and Vuser machines. Use a ping utility or type ping <server_name> from the DOS command line to verify communication with a remote machine. Make sure that the remote load generator and Controller machines can ping each other by IP addresses and machine names.

If the ping does not respond, or fails with a timeout, then the machine name is not recognized. To solve this problem, edit the hosts file located in the **WINNT\system32\drivers\etc** directory, and add a line with both the IP address and the name.

For example:

#	102.54.94.97	rhino.acme.com	# source server
#	38.25.63.10	x.acme.com	# x client host

Load Generator Connections

To verify the load generator connectivity, connect to each one of the remote load generators from the Controller's Load Generators dialog box. In the load generator's Platform field, select a Windows or UNIX platform. Select the load generators and click **Connect**. The status changes to "Connecting".

If the Connection fails, the status changes to "Failed" and details are written to the Details box. Double-click the Details box for more information about a failure.

If a connection succeeds, the status changes to "Ready", and the actual platform name appears in the Platform box (such as WINNT, UNIX, etc.).

Name	Status	Details
doc9pc	✔ Ready	
goose	✔ Ready	
miro	✔ Ready	
rman	✖ Failed	Connection to host failed. Communication problem: RPC: F
oxygen	✖ Failed	Connection to host failed. Communication problem: RPC: F
jukebox	✖ Failed	Connection to host failed. Communication problem: RPC: F
hammer	✖ Failed	Connection to host failed. Communication problem: RPC: F
steel	⇅ Connecting	Connection started.

If your scenario uses several domains (for example, Vusers on a different domain than the Controller), the Controller may have trouble communicating with the load generators. This occurs because the Controller uses the short load generator name—not including the domain—by default. To solve this, you must tell the Controller to determine the full load generator names, including the domains.

Modify the **miccomm.ini** file in the Controller machine's Windows directory as follows:

[tcpnet]
LocalHostNameType= 1

The possible values for LocalHostNameType are:

0 - Attempt to use the full machine name.
1 - Use the short machine name. This is the default.

Note: In certain environments such as WINS, load generators are unable to resolve machine names.

467

Connecting to a Controller with Multiple IP Addresses

If the load generator does not recognize the Controller machine by its short name or full name, and the Controller machine has more than one IP address, you can define an alias name for the Controller machine in the load generator's **hosts** file, located in the WINNT\system32\drivers\etc directory. The alias name should point to the IP address you want the load generator to recognize. For example: 255.0.0.1 delta.

UNIX Shell

For UNIX Vusers, make sure that the Windows Controller can execute a remote shell command. Type the following at the DOS command prompt: rsh -l <UNIX user login name> <load generator name> <command>. If you get a message indicating that permission is denied, make sure the **.rhosts** file in your UNIX home directory contains Controller machine permission for the user login name. In some cases, a "+" character must be added at the end of the **.rhosts** file. For example, if you log on to the Controller as **bill** and connect to the UNIX load generator as **mike**, you must ensure that **mike** allows **bill** to log on using his name. This can be done by adding the line "+ bill" at the beginning of mike's **.rhosts** file.

For more information on setting user login names, see "Configuring Load Generator Settings" on page 88.

To use UNIX without RSH:
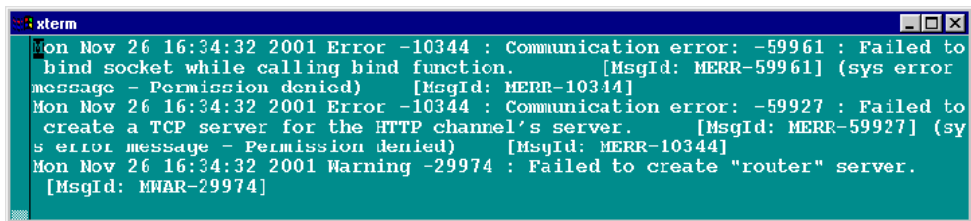
1 On the UNIX load generator, run the agent daemon by running the following command from <**LoadRunner directory**>/**bin**:

m_daemon_setup -install

This runs a daemon called m_agent_daemon, and if successful you will receive a message: m_agent_daemon installed successfully.

The agent will now keep running, even if the user is logged off. It will only stop running using the command explained in step 3 below, or by rebooting the machine.

➤ If you receive the message ERROR: File m_agent_daemon doesn't exist, this means that you are not in the same directory as the file (meaning not in **<LR_root>**/**bin** directory, or the file really doesn't exist, which indicates a problem with the installation).

➤ If a daemon of this name is already being run by the same user, you will receive the following warning:
WARNING: Could not install m_agent_daemon, reason - user <user_name> is already running m_agent_daemon on this machine.

➤ If an error occurred, you will receive the following error message:
ERROR: Could not install m_agent_daemon. Check log file m_agent_daemon[xxx].log in your temp directory.

➤ If you look at the log file m_agent_daemon[xxx].log in the temp directory, you will see the following errors, even if the installation succeeded:

These messages appear because the LoadRunner agent always tries to open port number 443 (because any agent can be a MI Listener, and the MI Listener always listens to this port), and in UNIX machines, this port cannot be opened by any user except for the root user. However, this will not interfere with using this agent for the load generator.

2 In the Controller, in the **Generators** > **Load Generator Information** > **UNIX Environment** tab, select the **Don't use RSH** option. Then connect as usual.

3 To stop the agent daemon, run the following command the **<LR_root>/bin** directory: m_daemon_setup -remove

This stops the m_agent_daemon, and if successful, you will receive a message: m_agent_daemon removed successfully.

➤ If no daemon of this name is being run by this user, you will receive the following warning:
WARNING: Could not remove m_agent_daemon, reason - user <user_name> is not running m_agent_daemon on this machine.

➤ If an error occurred, you will receive the following error message:
ERROR: Could not remove m_agent_daemon. Check log file m_agent_daemon[xxx].log in your temp directory.

Failure to Connect to the AUT Database

If you are running a database application, you must ensure that all remote clients can connect with the database server. If network or configuration errors occur when the client accesses the server, you must correct them before running a scenario. To ensure that your client application can connect with the database server, perform the following tests:

➤ **Ping.** Ensure that the client can communicate with the database server using TCP/IP. Use a ping utility or type ping <server_name> from the DOS command line.

➤ **SQL Utilities.** Use a simple utility such as ISQL or SQLPLUS to log on to the database server and perform several basic operations.

Failure to Access Files

A LoadRunner scenario will fail if the result path or Vuser script is inaccessible to one or more of the participating machines. Check the following items:

➤ Path Translation

➤ Vuser Script

➤ Result Path

Path Translation: A script's location (path) is always based on the Controller machine's mapping of that location. If a Vuser load generator maps to the script's path using a different name, path translation is required. Path translation translates the Controller's mapping of a given location to the Vuser load generator's mapping. For example, if one machine maps the script directory as **g:\test**, while another maps it as **h:\test**, the paths should be translated.

Path translation is also effective across platforms—between Windows and UNIX. You use path translation to translate the Windows Controller paths into paths recognized by UNIX.

Note: Path translation is required only if you chose to save all scripts and results to a shared network drive. In the default setup, LoadRunner saves files locally and collates them to the Controller machine; no path translation is required.

Suppose that your script is in the **/usr/jon/lr_test1** directory and runs on the UNIX machine, **sunny**. To translate it from the Windows Controller machine, **pc1**, where your UNIX directory is mapped as **r**, enter the following line in the path translation table:

| pc1 | r:\ | /usr/jon | sunny |

To translate the **f:\qa** Controller directory to all load generators running **/m/qa/lr_test2/lr_test2.usr** on a UNIX platform, type:

| win | f:\qa | /m/qa | UNIX |

If the paths are not translated properly, the scenario will fail. For more information about path translation, see Appendix B, "Performing Path Translation."

➤ **Vuser Script.** Make sure that the Vuser script is accessible to all load generators participating in the scenario through path translation and permissions. View or run the Vuser script in stand-alone mode on each of the participating load generators.

➤ **Result Path.** Make sure that the result path is accessible to all load generators participating in the scenario through path translation and permissions. Check the permissions of the result directory files and modify them if necessary.

Failed Vusers or Transactions

LoadRunner Vusers or transactions may fail for a variety of reasons relating to the network, database, or actual script. You can find information about scenario runs from the following sources:

➤ Run View

➤ Output Window

➤ Output File (excluding GUI Vusers)

➤ Analysis Reports and Graphs

Run View

The Run view is part of the Controller. The Scenario Groups pane in the upper-left corner of the view indicates the status of the Vuser groups during and after the scenario run. During the scenario run, the columns will show a **Pending**, **Initializing**, **Ready**, **Running**, or **Rendezvous** status. You can also view the status of each individual Vuser in the Vusers dialog box. If a Vuser fails and does not complete the script execution, LoadRunner displays an **error** status. If a Vuser completes the script execution, LoadRunner indicates the transaction status of a completed script run using the **Done.Failed** or **Done.Passed** status.

For more information about the Vuser states, see Chapter 16, "Viewing Vusers During Execution."

Output Window

View the Output window from the Controller. The output window contains useful information for debugging a scenario. The output window lists five types of messages: errors, warnings, notifications, debug, and batch. An error message usually results in a failed script. A warning message indicates that the Vuser encountered a problem, but test execution continued. A notification provides useful information such as recorded think time values and other run-time information. A debug message is sent if you enable the debugging feature in **Tools** > **Options** > **Debug Information** (Expert Mode). Batch messages are sent instead of message boxes appearing in the Controller, if you are using automation.
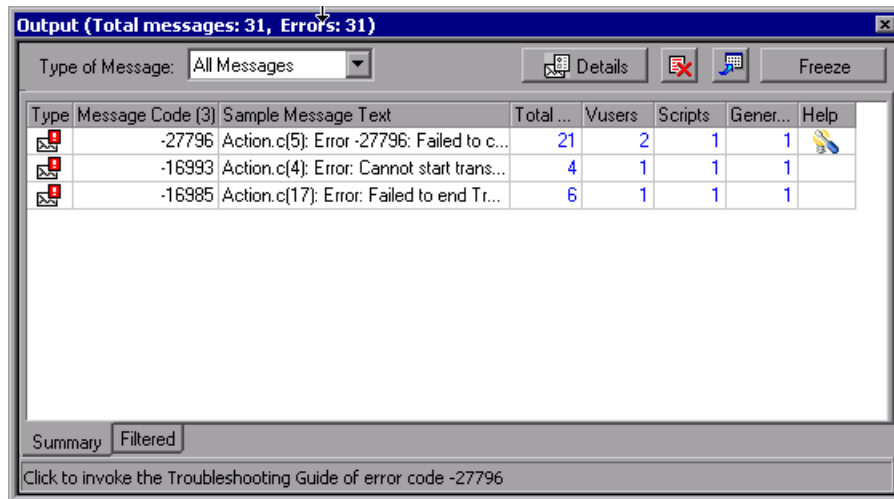
For more information about the Output window, see Chapter 16, "Viewing Vusers During Execution."

Output File

You can view information about script execution in an output file located in the Vuser results directory. The output file, **output.txt**, contains:

➤ a list of the primary functions called during the scenario

➤ error messages from the database server

➤ transactions and rendezvous information

The extent of the information sent to the output file depends on the output file settings. In the VuGen's run-time settings, you specify a Brief or Extended log. For the Extended log, you can specify a full trace, returned data, or current parameter value. An extended log is helpful for debugging a script, but if you are not debugging, Extended log is not recommended as it introduces extra overhead. For more information about configuring run-time settings, see the *HP Virtual User Generator User's Guide*.

Analysis Reports and Graphs

You can generate graphs and reports to view information about the scenario run. For example, the Summary report displays a table containing the scenario's run-time data and provides links to the following graphs: Running Vusers, Throughput (Web), Hits per Second (Web), HTTP Responses per Second, Transaction Summary, Total Errors, and Average Transaction Response Time.

For more information on the available graphs and reports, see the *HP LoadRunner Analysis User's Guide*.

Increasing the Number of Vusers on a Windows Machine

Under the normal settings of a Windows machine, you are limited to several hundred Vusers. This limitation is related to the operating system and not to the CPU or memory.

To work around the limitation of the Windows operating system, modify the Windows kernel as follows:

 1 Save a copy of the registry file in case you have trouble with these modifications.

 2 Run Regedit.

 3 Go to following key in KEY_LOCAL_MACHINE:

System\CurrentControlSet\Control\Session Manager\SubSystems

 4 Select the Windows key. The default Windows key for NT 4.0 looks like this:

```
%SystemRoot%\system32\csrss.exe ObjectDirectory=\Windows
SharedSection=1024,3072
Windows=On SubSystemType=Windows ServerDll=basesrv,1
ServerDll=winsrv:UserServerDllInitialization,3
ServerDll=winsrv:ConServerDllInitialization,2
ProfileControl=Off MaxRequestThreads=16
```

The SharedSection=1024,3072 key has the format xxxx,yyyy where:

xxxx defines the maximum size of the system-wide heap (in KB)

yyyy defines the size of the per desktop heap.

 5 Increase the SharedSection parameter by changing the yyyy settings from 3072 to 8192 (which is 8 MB).

This setup successfully allowed 1250 Oracle Vusers to run on a Windows machine using 2 Pentium PRO 200 MHz with 1 GB RAM.

Each Vuser in this setup used approximately 2MB memory. Other Vusers may require more memory.

LoadRunner was able to load over 2500 Vusers when the Windows terminal server was run as the Operating System and the above registry setting was changed.

The above registry changes enable you to run more threads, allowing you to run more Vusers on the machine. This implies that you are not bound by the Windows operating system; you are only bound by hardware and internal scalability limitations.

Troubleshooting Firewalls

There are three log files that provide additional information about activity over the firewall.

The **LoadRunner agent log file** contains information about communication activity between the LoadRunner agent and the MI Listener.

➤ To open the file on Windows machines, right-click the LoadRunner agent icon in the system tray of the LoadRunner agent machine, and select **View Log**. Alternatively, open the latest **<temp_directory>\LoadRunner_agent_startup<unique identifier>.log** file (if the LoadRunner agent is a process), or **<temp_directory>\LoadRunner_agent_service<unique identifier>.log** file (if the LoadRunner agent is a service) in a text editor.

➤ To open the file on UNIX machines, open the **<temp_directory>/m_agent_daemon<unique identifier>.log** file in a text editor.

The **MI Listener log file** contains information about MI Listener communication with the LoadRunner agent and the Controller.

To open the file, right-click the MI Listener Agent icon in the system tray of the MI Listener machine, and select **View Log**. Alternatively, open the latest **<temp_directory>\LoadRunner_agent_startup<unique identifier>.log** file (if the LoadRunner agent is a process), or **<temp_directory>\LoadRunner_agent_service<unique identifier>.log** file (if the LoadRunner agent is a service) in a text editor.

The **Controller log file** contains information about communication activity between the Controller and the MI Listener.

To open the file on Windows machines, open the **<temp_directory>\drv_log.txt** file in a text editor.

Verifying Connection Between LoadRunner Agent and MI Listener

If there is a proper connection between the LoadRunner agent and the MI Listener:

➤ On Windows platforms, the agent icon's light in the system tray will turn from red to green.

➤ On UNIX platforms, a file called **<Local_machine_key>_connected_to_MI_Listener** will be created in the temporary directory of the LoadRunner agent machine. Local_machine_key is the value set in the Agent Configuration, as described in Chapter 17, "Working with Firewalls in LoadRunner." The file will be removed when the LoadRunner agent disconnects from the MI Listener.

➤ On both UNIX and Windows platforms, the following message will appear in the LoadRunner agent log file: Notify Connected to MI Listener.

Note: The LoadRunner agent tries to connect to the MI Listener machine every Timeout seconds (as defined in the Agent Configuration). After a successful connection, if no Controller has connected through this MI Listener to the agent after another Timeout, LoadRunner will disconnect from the Controller.

On a Windows machine, the agent icon's light in the system tray will turn from green to red. On UNIX machines, the file <Local_machine_key>_connected_to_MI_Listener will be removed from the temporary directory in the LoadRunner agent machine.

In both Windows and UNIX, the message Disconnected from MI Listener will appear in the LoadRunner agent log file.

UNIX Connection Errors

After installing the **m_agent_daemon** as described in Chapter 17, "Working with Firewalls in LoadRunner." you should receive a message: m_agent_daemon installed successfully.

Agent Daemon Errors

ERROR: File m_agent_daemon doesn't exist.

This error means that you are not in the same directory as the file (meaning not in **<LR_root>/bin** directory, or the file really doesn't exist, which indicates a problem with the installation).

WARNING: Could not install m_agent_daemon, reason - user <user_name> is already running m_agent_daemon on this machine.

This warning message occurs when a daemon of this name is already being run by the same user.

ERROR: Could not install m_agent_daemon. Check log file m_agent_daemon[xxx].log in your temp directory.

This error indicates that some error has occurred when loading the daemon. You should check the log file and consult the following troubleshooting tips.

LoadRunner Agent Log File Errors

Error - 10344 : Communication Error: -59961 : Failed to bind a socket while calling bind function.

Error -10344 : Communication Error: -59927 : Failed to create a TCP server for the HTTP channel's server.

Warning -29974 : Failed to create "router" server.

These messages appear because the LoadRunner agent always tries to open port number 443 (because any agent can be a MI Listener, and the MI Listener always listens to this port), and in UNIX machines, this port cannot be opened by any user except for the root user. However, this will not interfere with using this agent for the load generator.

Error -10343 : Communication error : -59981 : Failed to connect to remote host - <MI_Listener_name>.

The MI Listener is not being run at the time of the connection attempt on the machine set in MI Listener Name in the Agent Configuration.

Error -10343 : Communication error: -59928 : Unresolved server name.

The name passed in MI Listener Name in the Agent Configuration is not a name, full name, or IP address of a valid machine, or no value was set.

Error -10343 : Communication error: -59928 : Unresolved server name.

The name passed in Proxy Name in the Agent Configuration is not a name, full name, or IP address of a valid machine.

Error -10343 : Communication error: -59945 : Client failed to connect to a PROXY Server with the following settings: (-server_port=<proxy_server_port>)(-server_fd_primary=2)(-server_type=8)(-allowed_msg_size=0)(-allowed_msgs_num=0)(-proxy_configuration_on)(-tcp_tunnel_configuration_on).

The Proxy Name field is empty.

Error -10343 : Communication error: -59982 : Failed to connect to remote host - <MI_Listener_Name>. The remote address is not a valid address.

Error -10343 : Communication error: -59945 : Client failed to connect to a PROXY Server with the following settings: (-server_name=<proxy_server_name>)(-server_port=<proxy_server_port>)(-server_fd_primary=2)(-server_type=8)(-allowed_msg_size=0)(-allowed_msgs_num=0)(-proxy_configuration_on)(-tcp_tunnel_configuration_on).

The Proxy Port set in Agent Configuration has been set to the wrong port number.

Error -10343 : Communication error: -59913 : NTLM authentication to proxy server error - connection to proxy refused.

The proxy server is configured for NTLM authentication, and the Proxy User Name, Proxy Password and/or Proxy Domain are not set correctly in the Agent Configuration.

Error -10343 : Communication error: - 59880 : Basic authentication to proxy server error - connection to proxy refused.

The proxy server is configured in for Basic authentication and the Proxy User Name and/or Proxy Password are not set correctly in the Agent Configuration.

Error -10343 : Communication error: -59907 : SSL connect error : verify host failed : wrong DNS test.

This error occurs when you have set the Check Server Certificates setting to True, and have not issued a new certificate to the MI Listener machine (see Appendix G, "Working with Digital Certificates" for more details).

Error -10343 : Communication error: -59907 : SSL connect error : certificate verify failed.

Error -10343 : Communication error: -59907 : SSL connect error : sslv3 alert handshake failure.

Error -10343 : Communication error: -59907 : SSL connect error : sslv3 alert bad certificate.

Error -10343 : Communication error: -59907 : SSL connect error : sslv3 alert certificate expired.

These errors occur when you set the Check Server Certificates setting to True. See Appendix G, "Working with Digital Certificates" to learn how to issue a valid certificate.

Error -10343 : Communication error: -59910 : SSL initialization error : Certificate not found.

Error -10343 : Communication error: -59910 : SSL initialization error : No such file or directory.

Error -10343 : Communication error: -59910 : SSL initialization error : system lib.

These errors occur when the Client Certificate owner setting in the Agent Configuration is set to True, but no certificate was installed in the LoadRunner agent machine (see Appendix G, "Working with Digital Certificates" for more details).

MI Listener Log File Errors

Error - 10344 : Communication Error: -59961 : Failed to bind a socket while calling bind function.

Error -10344 : Communication Error: -59927 : Failed to create a TCP server for the HTTP channel's server.

Warning -29974 : Failed to create "router" server.

This error means that another process on the MI Listener machine is occupying port 443 (for instance, the IIS service).

Error -10343 : Communication error: -59904 : SSL accept error : sslv3 alert certificate expired.

These errors occur when you have set the Check Server Certificates setting to True, and the MI Listener's certificate is expired.

Error -10343 : Communication error: -59904 : SSL accept error : sslv3 alert bad certificate.

These errors occur when you have set the Check Server Certificates setting to True, and either:

➤ the MI Listener's certificate does not have a signature that is included in the LoadRunner agent's CA List

➤ the MI Listener's certificate has a future verification date

See Appendix G, "Working with Digital Certificates" to learn how to issue a valid certificate and how to add a Certification Authority to a CA list, or how to create a certificate with a new validation date.

Error -10343 : Communication error: -59904 : SSL accept error : peer did not return a certificate.

These errors indicate that the Check Client Certificates setting in the MI Listener Configuration is set to True, but the Client Certificate owner setting in the Agent Configuration is set to False.

Error -10343 : Communication error: -59904 : SSL accept error : no certificate returned.

These errors indicate that the Check Client Certificates setting in the MI Listener Configuration is set to True, and the Client Certificate owner setting in the Agent Configuration is set to True, but either:

➤ the LoadRunner agent's certificate does not have a signature that is included in the MI Listener's CA List

➤ the LoadRunner agent's certificate has a future verification date.

See Appendix G, "Working with Digital Certificates" to learn how to issue a valid certificate and how to add a Certification Authority to a CA list, or how to create a certificate with a new validation date

Error -10343 : Communication error: -59904 : SSL accept error : no certificate returned.

These errors indicate that the Check Client Certificates setting in the MI Listener Configuration is set to True, and the Client Certificate Owner setting in the Agent Configuration is set to True, but the LoadRunner agent's certificate has expired.

General Connection Errors

These errors can occur when using all configurations.

If no errors appear both in the LoadRunner agent log and the MI Listener log, but the agent does not connect to the MI Listener, make sure that the FireWallServiceActive attribute in the Firewall section in the **<LR_Installation>\dat\br_lnch_server.cfg** file on the LoadRunner agent machine, is set to 1.

Verifying Connection Between the Controller and Agent through the MI Listener

When there is a successful connection between the LoadRunner agent and the MI Listener, and the Controller machine fails to connect, you should check the following:

➤ The **Name** field in the Load Generators dialog in the Controller should match the name set in the **Local Machine Key** in the Agent Configuration.

➤ The **MI Listener** box in the **Load Generators** > **Details** > **Security** tab of the above host matches the name set in the **MI Listener Name** in the Agent Configuration.

➤ In the **Tools** menu of the Controller, in the **Options** > **Timeout** tab, the **Load Generator Connect timeout** might need to be increased because the Firewalls may slow down the communication.

➤ Make sure that the Controller machine recognizes the LoadRunner agent machine (e.g., by using the ping utility). If this fails, there is a configuration problem in the system not related to LoadRunner, and it must be solved before the connection can be made.

➤ Make sure that the Controller has successfully connected to the MI Listener by checking port 50500 on the MI Listener machine (you can use the netstat utility on the MI Listener machine).

Working with the LoadRunner Agent

The LoadRunner Agent runs on the load generators and enables communication between the Controller, load generators, and MI Listeners (in firewall configurations). The agent receives instructions from the Controller to initialize, run, pause, and stop Vusers. At the same time, the agent also relays data on the status of the Vusers back to the Controller.

To check the agent's current configuration, move your mouse over the Agent's icon in the Task bar area, and read the description. The description will say either "LoadRunner Agent Process" or "LoadRunner Agent Service."

Running the LoadRunner Agent as a Process

In some cases, such as SAPGUI replay, running GUI Vusers on remote machines, or Terminal sessions, the LoadRunner Agent must run as a process.

To change the LoadRunner Agent from a service to a process:

1 To uninstall the service, run
   <LR_dir>**launch_service****bin****magentservice.exe -remove**

2 To install the process, make a shortcut of
   <LR_dir>**launch_service****bin****magentproc.exe** and place it in your system startup folder.

3 To run the process, run **magentproc.exe** from the startup folder or from
   <LR_dir>**launch_service****bin**, or restart your machine.

Running the LoadRunner Agent as a Service

In most cases, the LoadRunner Agent runs as a service.

To change the LoadRunner Agent from a process to a service:

1 Stop the agent process by right-clicking its icon in the system tray and selecting **Close**.

2 To uninstall the process, delete its shortcut from the system startup folder.

3 To install the service, run *<LR_dir>***launch_service\bin\magentservice.exe -install** *<user_domain>\<user_name> <password>*

This also runs the agent service.

In the Services dialog the LoadRunner Agent Service will run under the name **Local System**, so that it can be interactive with the Desktop. However, it still uses the supplied user credentials. All of the Agent's child processes (such as **lr_bridge.exe** and **mdrv.exe**) will run under the supplied user credentials.

4 To run the service later, after stopping it, start it from the Services dialog box.

Changing the LoadRunner Agent Password

To change the password of the LoadRunner Agent service you first need to uninstall the service, and then reinstall it with the new user name and password.

1 To uninstall the service, run
<LR_dir>**launch_service\bin\magentservice.exe -remove**

2 To reinstall the service, run
<LR_dir>**launch_service\bin\magentservice.exe -install**
<user_domain>\<user_name> <password>

This command also runs the agent service.

Do not change the user name or password in the services dialog.

Mapping Network Drives when Running the Agent as a Service

For all Windows platforms, when the user is logged out, the service cannot resolve the mapping of network drives. For Windows XP, the service also cannot resolve the mapping of network drives when the user is logged on.

In cases when the service cannot work with mapped network drives, use the full path to the directory, for example, <\\<machine-name>\<directory>\> .

E

Configuring Multiple IP Addresses

When you run a scenario, the Vusers on each load generator use the machine's IP address. You can define multiple IP addresses on a load generator to emulate a real-life situation in which users sit on different machines.

This chapter includes:

About Multiple IP Addresses

Application servers and network devices use IP addresses to identify clients. The application server often caches information about clients coming from the same machine. Network routers try to cache source and destination information to optimize throughput. If many users have the same IP address, both the server and the routers try to optimize. Since Vusers on the same load generator have the same IP address, server and router optimizations do not reflect real-life situations.

's multiple IP address feature enables Vusers running on a single machine to be identified by many IP addresses. The server and router recognize the Vusers as coming from different machines and as a result, the testing environment is more realistic.

Applicable Protocols

The multiple IP address feature is applicable to the following protocols:

➤ **Client/Server.** DNS, Windows Sockets

➤ **Custom.** Javascript Vuser, VB Vuser, VB Script Vuser

➤ **E-business.** FTP, Palm, SOAP, Web (HTTP/HTML), Web Services, WinSock\Web Dual Protocol

➤ **ERP/CRM.** Oracle NCA, Oracle Web Applications 11i, PeopleSoft Enterprise, SAP-Web, Siebel-Web

➤ **Legacy.** RTE

➤ **Mailing Services.** Internet Messaging (IMAP), POP3, SMTP

➤ **Streaming Data.** Real

➤ **Wireless.** i-Mode, VoiceXML, WAP

This feature can be implemented on Windows and UNIX platforms.

Adding IP Addresses to a Load Generator

LoadRunner includes an IP Wizard program that you run on each Windows NT or Windows 2000 load generator to create multiple IP addresses. You add new IP addresses to a machine once and use the addresses for all scenarios. For information about adding IP addresses on UNIX machines, see "Configuring Multiple IP Addresses on UNIX" on page 494.

The following procedure summarizes how to add new IP addresses to a load generator:

1 Run the IP Wizard on the load generator to add a specified number of IP addresses. Manually configure the new IP addresses for UNIX load generators.

2 Restart the machine.

3 Update the server's routing table with the new addresses, if necessary.

4 Enable this feature from the Controller. See "Enabling Multiple IP Addressing from the Controller" on page 497.

Using the IP Wizard

The IP Wizard resides on each load generator. You run this process once to create and save new IP addresses on Windows machines. The new addresses can be a range of addresses defined by the Internet Assignment Numbers Authority. They are for internal use only, and cannot connect to the Internet. This range of addresses is the default used by the IP Wizard.

To add new IP addresses to a load generator:

1 Invoke the IP Wizard from the LoadRunner program group.
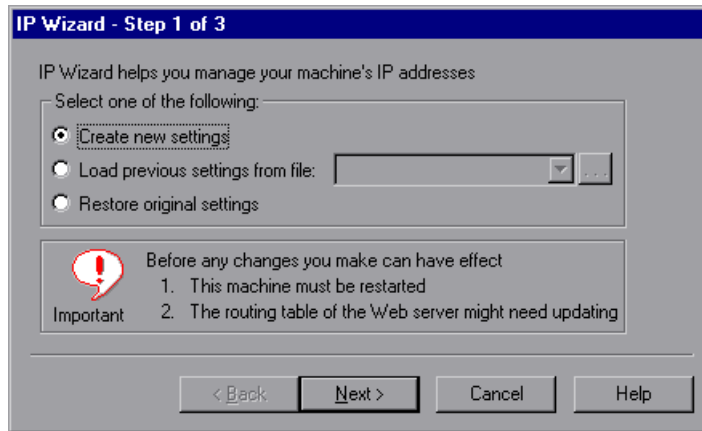
2 If you have an existing file with IP address settings, select **Load previous settings from file** and choose the file.

3 If you are defining new settings, select **Create new settings**.

4 Click **Next** to proceed to the next step. If you have more than one network card, choose the card to use for IP addresses and click **Next**.

The optional Web server IP address step enables the IP Wizard to check the server's routing table to see if it requires updating after the new IP addresses are added to the load generator.

```
IP Wizard - Step 2 of 3

    Optional:  You can enter your Web server's IP address here.

    IP Wizard will check if the Web server's routing table will need updating.
    If so, scripts will be generated to help you update.


                        ┌──────────────────────┐
                        │   .      .      .     │
                        └──────────────────────┘



              ┌─────────┐ ┌─────────┐  ┌─────────┐   ┌─────────┐
              │ < Back  │ │ Next >  │  │ Cancel  │   │  Help   │
              └─────────┘ └─────────┘  └─────────┘   └─────────┘
```
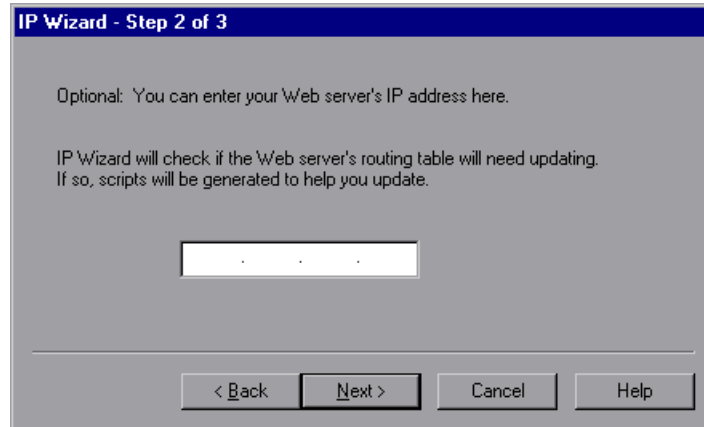
5 To check the server's routing table directly after adding the addresses, enter the server IP address. See "Updating the Routing Table" on page 496 for more information.

6 Click **Next** to see a list of the machine's IP address(es). Click **Add** to define the range of addresses.
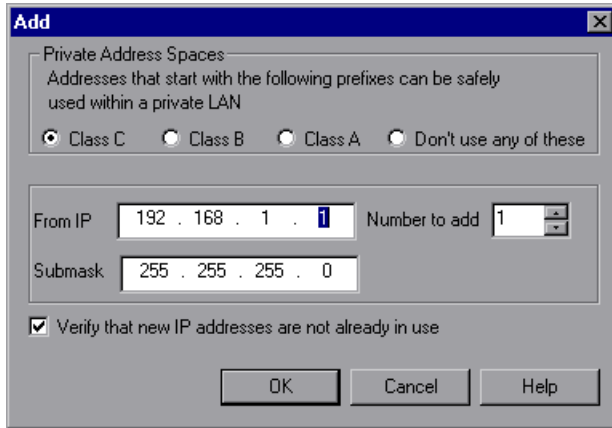
IP addresses include two components, a **netid** and **hostid**. The submask determines where the netid portion of the address stops and where the hostid begins.

7 Select a class that represents the correct submask for the machine's IP addresses.

8 Specify the number of addresses to create. Select **Verify that new IP addresses are not already in use** to instruct the IP Wizard to check the new addresses. The IP Wizard will only add the addresses not in use.

Click **OK** to proceed.

9 After the IP Wizard creates the new addresses, the summary dialog box lists all of the IP addresses.

```
IP Wizard - Step 3 of 3

    IP Address          Subnet Mask
    207.232.012.025     255.255.255.000
    192.168.1.1         255.255.255.0
    192.168.1.2         255.255.255.0

                                         Number of
                                         IP's added:

                                              2

    [  Add...  ]    [  Remove  ]

              [ < Back ]  [ Finish ]  [ Cancel ]  [ Help ]
```
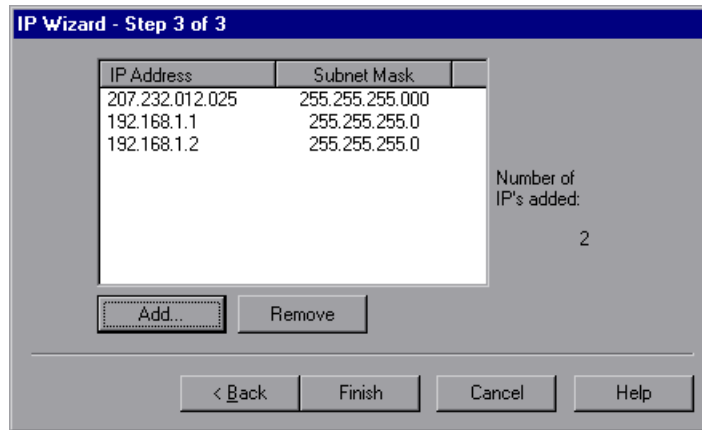
10 Click **Finish** to exit the IP Wizard. The IP Wizard Summary dialog box is displayed.

```
IP Wizard - Summary                                        [x]

    The following scripts have been generated to help you add IP
    addresses to the routing table of Web server 200.200.200.200

            E:\TEMP\unix_routing.sh
            E:\TEMP\nt_routing.bat

    IP Wizard will add the following IP addresses
    to this machine:

      192.168.1.1  Mask 255.255.255.0


    [ ] Reboot now to update routing tables


    [ Save ]  [ Save as... ]  [  OK  ]  [ Cancel ]
```
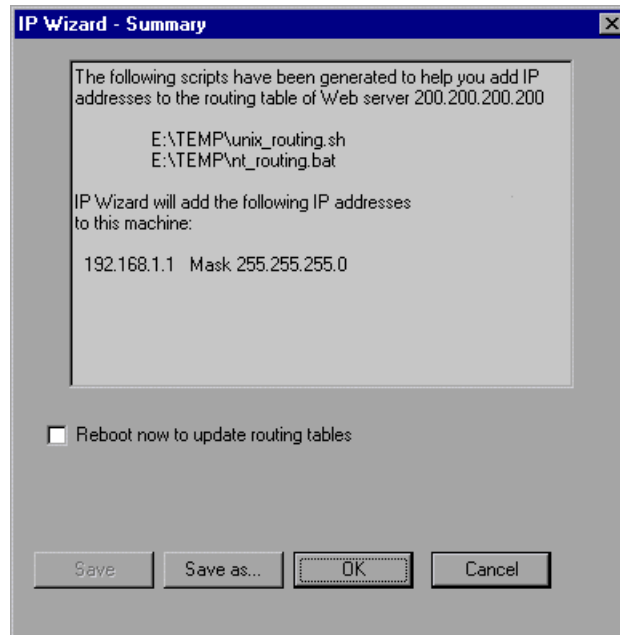
11 Note the address of the **.bat** file, and see "Updating the Routing Table" on page 496 for information about using the batch file to update the routing table, if necessary.

12 After you update the routing table, check **Reboot now to update routing tables** to initialize the NT device drivers with the new addresses.

13 Click **OK**.

Configuring Multiple IP Addresses on UNIX

To configure multiple IP addresses on UNIX, manually configure the addresses on the load generator.

➤ Solaris 2.5, 2.6, 7.0, 8.0

➤ Linux

➤ HP 11.0 or higher

➤ IBM AIX

Solaris 2.5, 2.6, 7.0, 8.0

To configure the hme0 device to support more than one IP address:

1 Create entries in **/etc/hosts** for each hostname on your physical machine:

128.195.10.31 myhost
128.195.10.46 myhost2
128.195.10.78 myhost3

Create **/etc/hostname.hme0:n** files that contain the hostname for the virtual host **n**.

Note: hostname.hme0:0 is the same as **hostname.hme0**.

/etc/hostname.hme0   (Contains name myhost)

/etc/hostname.hme0:1 (Contains name myhost2)

/etc/hostname.hme0:2 (Contains name myhost3)

The above changes will cause the virtual hosts to be configured at boot time.

2 You can also directly enable/modify a logical hosts configuration by running **ifconfig** directly on one of the logical hosts, using the **hme0:n** naming scheme:

```
% ifconfig hme0:1 up
% ifconfig hme0:1 129.153.76.72
% ifconfig hme0:1 down
```

To verify the current configuration, use **ifconfig –a**.

Linux

To define multiple IP addresses for a single Ethernet card, you need IP Aliasing compiled into the kernel. To do this, use the **ifconfig** command:

```
/sbin/ifconfig eth0:0 x.x.x.x netmask 255.255.x.x up
```

Substitute the new IP address for x.x.x.x, and insert the correct information for subnet mask. Place this command in the **rc.local** file so that it executes upon boot.

HP 11.0 or higher

To define multiple IP addresses for a single Ethernet card, you need IP Aliasing compiled into the kernel. To do this, use the **ifconfig** command:

```
/sbin/ifconfig lan1:0 x.x.x.x netmask 255.255.x.x up
```

Substitute the new IP address for x.x.x.x, and insert the correct information for subnet mask. Place this command in the **rc.local** file so that it executes upon boot.

IBM AIX

To define multiple IP addresses for a single Ethernet card, you need IP Aliasing compiled into the kernel. To do this, use the **ifconfig** command:

/usr/sbin/ifconfig [int] [ip address] alias netmask [mask]

For example, if you want to add IP address 10.0.0.1 to the main interface, you need to run, as root, the following:

/usr/sbin/ifconfig ne0 10.0.0.1 alias netmask 255.255.255.0

To execute this line upon boot, create a standard script in the appropriate run level (**/etc/rc.d/rc#.d**).

Updating the Routing Table

Once the client machine has new IP addresses, the server needs the addresses in its routing table, so that it can recognize the route back to the client. If the server and client share the same netmask, IP class, and network, the server's routing table does not require modification.

Note: If there is a router between the client and server machines, the server needs to recognize the path via the router. Make sure to add the following to the server routing table: route from the Web server to the router, and routes from the router to all of the IP addresses on the load generator.

To update the Web server routing table:

1 Edit the batch file that appears in the IP Wizard Summary screen. An example **.bat** file is shown below.

```
REM This is a bat file to add IP addresses to the routing table of a
server
REM Replace [CLIENT_IP] with the IP of this machine that the server
already recognizes
REM This script should be executed on the server machine

route ADD 192.168.1.50 MASK 255.255.255.255 [CLIENT_IP] METRIC 1
route ADD 192.168.1.51 MASK 255.255.255.255 [CLIENT_IP] METRIC 1
route ADD 192.168.1.52 MASK 255.255.255.255 [CLIENT_IP] METRIC 1
route ADD 192.168.1.53 MASK 255.255.255.255 [CLIENT_IP] METRIC 1
route ADD 192.168.1.54 MASK 255.255.255.255 [CLIENT_IP] METRIC 1
```

2 For each occurrence of **[CLIENT_IP]**, insert your IP address instead.

3 Run the batch file on the server machine.

Enabling Multiple IP Addressing from the Controller

Once you define multiple IP addresses, you set an option to tell the Controller to use this feature.

To enable multiple IP addressing from the Controller:

1 In the Controller Design view, select **Scenario > Enable IP Spoofer**.

Note: You must select this option before connecting to a load generator.

---

2 Use the **General Options** of the Controller Expert Mode to specify how the Controller should implement this feature.

For more information, see Appendix C, "Working in Expert Mode."

F

Controller Command Line Arguments

When you invoke the Controller from the command line, you can pass arguments to instruct the Controller how to behave. By passing arguments in the command line, you configure Controller scenario settings without the need to manually define them using the Controller UI.

This chapter includes:

About Controller Command Line Arguments

When invoked, the Controller checks all of the received arguments and sets its start-up environment accordingly. If no arguments are passed, the Controller uses its default settings.

For example, you can instruct Controller to Connect to Quality Center on start-up, save results to a directory other than the directory defined in the scenario, and invoke Analysis upon scenario termination.

Invoking the Controller from the Command Line

To invoke the Controller, type wlrun in the command line, followed by the arguments. Each argument should be preceded by a dash.

Note: The arguments are case-sensitive.

For example:

wlrun -TestPath C:\LoadRunner\scenario\Scenario.lrs -Run

When you invoke the Controller from the command line, the following rules apply:

➤ If the Controller is invoked with no arguments in the command line, the Controller uses its default settings.

➤ The Controller will always overwrite results.

➤ The Controller will automatically terminate upon scenario termination and results will be collected. If you don't want the Controller to automatically terminate upon scenario termination, add the flag -DontClose to the command line.

➤ The Controller launched through the command line behaves normally except when using the -Run option. Using the -Run option, dialogs and message boxes that usually open and require the user to close them in a usual launch, do not open in a command line launch.

➤ The Controller's settings are loaded from **wlrun5.ini**, located in Windows directory.

Quality Center Arguments

These arguments define the LoadRunner integration with Quality Center. For more information about the LoadRunner Quality Center integration, see Chapter 13, "Managing Scenarios Using Quality Center."

ConnectToQC	Specifies whether the Controller should connect to Quality Center on startup (**0/1** or **ON/OFF**)
QCServer	Quality Center server name. Must be a machine where Quality Center is installed
QCDB	Quality Center database name. Use the format: "<Domain name>.<Project name>".
UserName	User name for connecting to Quality Center
Password	Password corresponding to the user name
TestPath	Path to scenario in Quality Center database. For example, "[TD]\Subject\LoadRunner\Scenario1" If path includes blank spaces, use quotation marks.
TestId	Test ID (used by Quality Center only)
ResultCleanName	For use with **ResultCycle** only. Example: "Res1"
ResultCycle	Quality Center cycle. For example, "LR_60_SP1_247" **Note:** The **ResultCycle** and **ResultCleanName** arguments are required if you wish to store the results within the Quality Center database.

Run-Time Arguments

These arguments specify the run-time related scenario settings. For more information on scenario settings, see Chapter 12, "Preparing to Run a Scenario."

TestPath	Path to the scenario, for example, C:\LoadRunner\scenario\Scenario.lrs
	This argument can also be used for a scenario residing in a Quality Center database. For example, "[TD]\Subject\LoadRunner\Scenario1"
	If the path includes blank spaces, use quotation marks.
Run	Runs the scenario, dumps all output messages into **res_dir\output.txt** and closes Controller
InvokeAnalysis	Instructs LoadRunner to invoke Analysis upon scenario termination. If this argument is not specified, LoadRunner uses the scenario default setting.
ResultName	Full results path. For example, "C:\Temp\Res_01"
ResultCleanName	Results name. For example, "Res_01"
ResultLocation	Results directory. For example, "C:\Temp"

Note: If the scenario doesn't specify a results directory, and one of the results arguments was not passed, the scenario will not run.

G

Working with Digital Certificates

A Digital Certificate is an electronic "credit card" that establishes your credentials when doing business or other transactions on the Web. It is issued by a Certification Authority (CA). It contains the IP address of the machine for which it was issued, a validation date, and the digital signature of the certificate-issuing authority.

This chapter includes:

➤ Using Digital Certificates with Firewalls on page 503

➤ Creating and Using Digital Certificates on page 504

Using Digital Certificates with Firewalls

When the MI Listener sends its Public Key to the LoadRunner agent, it always sends its certificate as well (this is the server-side certificate). The LoadRunner agent can be configured to authenticate the certificate which it received, as described in Chapter 17, "Working with Firewalls in LoadRunner." If the agent is configured to authenticate the certificate, it can verify whether the sender is really the machine that it claims to be by:

➤ Comparing the certificate's IP address with the sender's IP address.

➤ Checking the validation date.

➤ Looking for the digital signature in its Certification Authorities list.

The MI Listener may also require the LoadRunner agent to send a certificate at any point in the session. This is called the client-side certificate, as described in the MI Listener Configuration Settings in Chapter 17, "Working with Firewalls in LoadRunner." If the LoadRunner agent owns a certificate, it sends it to the MI Listener for the same authentication process. If the LoadRunner agent does not own a certificate, the communication might not be continued.
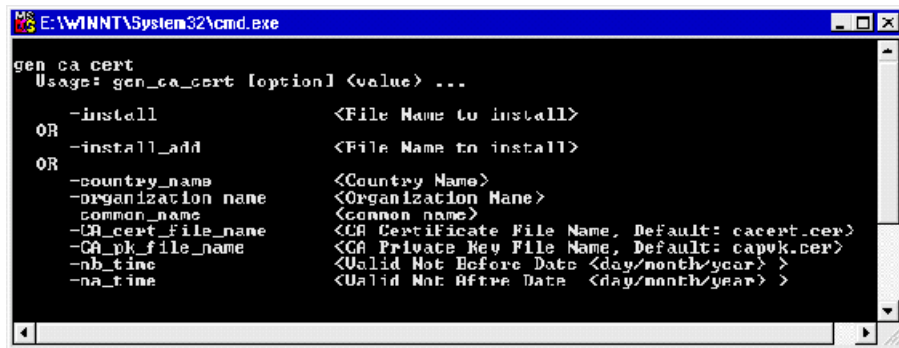
An SSL CA list and an SSL Certificate are included in each LoadRunner installation. This certificate is the same for all LoadRunner installations, which means that it can be obtained by third parties. Therefore, if you are interested in a more secure process, you should create your own Certificate Authority and include it in the list, and issue matching certificates for your machines.

Creating and Using Digital Certificates

You create a Certification Authority using the **gen_ca_cert.exe** (on UNIX platforms gen_ca_cert) utility, and a Digital Certificate using the **gen_cert.exe** (on UNIX platforms gen_cert) utility. Both utilities can be used on UNIX and Windows platforms, using a command-line interface.

To create a Certificate Authority using gen_ca_cert:

1 To view the format and usage, run the **gen_ca_cert** utility from the **<LoadRunner root folder>\launch_service\bin** directory.
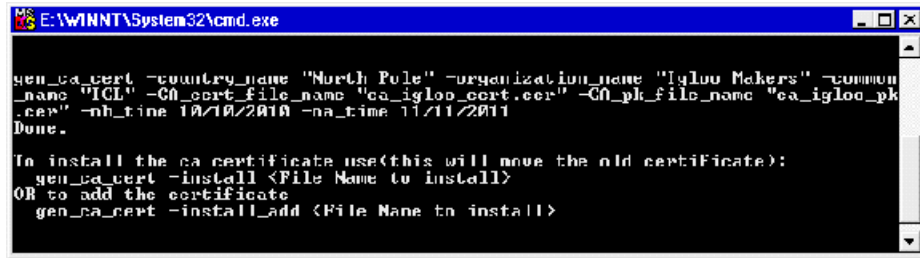
2 Create a new Certificate Authority by running the gen_ca_cert command with at least one of the options: -country_name <country name> -organization_name <organization name> and -common_name <the name of the CA>.

This process creates two files in the directory from which the utility was run: the CA Certificate (**cacert.cer**), and the CA Private Key (**capvk.cer**). To provide different file names, use the -CA_cert_file_name and the -CA_pk_file_name options respectively.

By default, the CA is valid for three years, from the time that the CA is generated. To change the validation dates, use the options -nb_time <beginning of validity in dd/mm/yyyy format> and/or -na_time <ending of validity in dd/mm/yyyy format>.

The following example creates two files: **ca_igloo_cert.cer** and **ca_igloo_pk.cer** in the current directory:

3 To install this CA, use the -install <name of certificate file> option. This option replaces any previous CA list and creates a new one that includes only this CA.

To add the new CA to the existing CA list, use the -install_add <name of certificate file>.
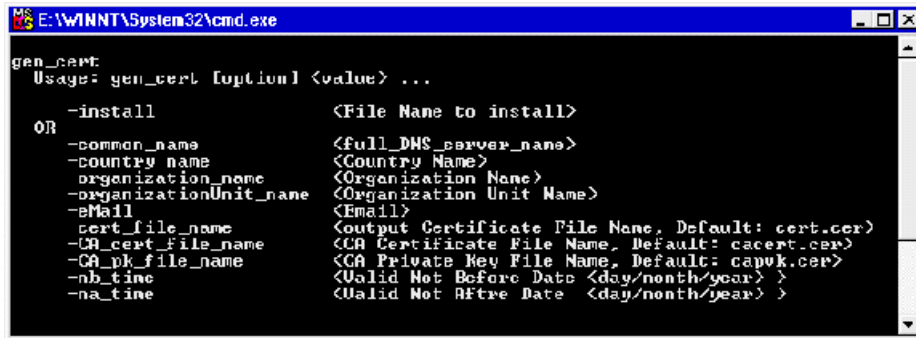
4 The -install and -install_add options install the certificate file only. Keep the private key file in a safe place and use it only for issuing certificates.

To create a Digital Certificate using gen_cert:

1 To view the format and usage, run the **gen_cert** utility from the <**LoadRunner root folder**>**launch_service****bin** directory.

2 Create a new Digital Certificate by running the gen_cert command with at least one of the options: -country_name <country name>, -organization_name <organization name>, -organization_unit_name <organization unit name>, -eMail <email address> and -common_name <the name, full name or IP address of the machine>.

The CA Certificate and the CA Private Key files are necessary for the creation of the certificate. By default, it is assumed that they are in the current directory, and are named **cacert.cer** and **capvk.cer** respectively. In any other case, use the -CA_cert_file_name and -CA_pk_file_name options to give the correct files and locations.

In this process, the certificate file is created in the directory from which the utility was run. By default, the file name is **cert.cer**. To provide a different name, use the -cert_file_name option.

By default, the CA is valid for three years, from the time that the CA is generated. To change the validation dates, use the -nb_time <beginning of validity in dd/mm/yyyy format> and/or -na_time <ending of validity in dd/mm/yyyy format> options .

The following example creates the **igloo_cert.cer** file in the current directory:

3 If you wish to install this certificate, use the -install <name of certificate file> option. This option replaces any previous certificate, as it is possible to own only one certificate per machine.

Index

Index