VMware Workbench Getting Started Guide

VMware Workbench 3.5.3

This document supports the version of each product listed and supports all subsequent versions until the document is replaced by a new edition. To check for more recent editions of this document, see http://www.vmware.com/support/pubs.

vmware®

You can find the most up-to-date technical documentation on the VMware Web site at:

http://www.vmware.com/support/

The VMware Web site also provides the latest product updates.

If you have comments about this documentation, submit your feedback to:

docfeedback@vmware.com

Contents

About This Book

This book, the *VMware® Workbench Getting Started Guide,* provides information about VMware Workbench and its extensions to the Eclipse IDE.

Revision History

A revision of this book occurs with each release of the product, or as needed. A revised version can contain minor or major changes. Table 1 lists the revision history of this book.

Table 1. Revision History

Revision	Description
10 September 2015	vSphere 6.0 U1 GA – Workbench 3.5.3 GA
12 March 2015	vSphere 6.0 GA – Workbench 3.5.1 GA
29 Sept 2014	vSphere 6.0 RC – Workbench 3.5 GA
30 June 2014	vSphere 6.0 Beta 2 – Workbench 3.5 Beta

Intended Audience

This book is for VMware customers partners who develop and certify products using VMware Workbench.

VMware Technical Publications Glossary

VMware Technical Publications provides a glossary of terms that might be unfamiliar to you. For definitions of terms as they are used in VMware technical documentation, go to http://www.vmware.com/support/pubs.

Documentation Resources

Each Workbench plug-in contains its own help documentation that displays in the **Help > Help Contents** window. PDF copies of these documents may be available on the VMware Developer Center Web site.

Documentation Feedback

Please send feedback to this documentation through your VMware partners representative.

VMware also welcomes your suggestions for improving our other VMware API and SDK documentation. Send your feedback to: docfeedback@vmware.com.

Technical Support and Education Resources

For all questions relating specifically to the VMware Workbench, see the VMware partner co-dev web site:

http://www.vmware.com/partners/alliances/programs/co-dev/iovp.htm

The following sections describe VMware technical support and educational resources available to you.

Online Support and Telephone Support

To use online support to submit technical support requests, view your product and contract information, and register your products, go to http://www.vmware.com/support.

Support Offerings

To find out how VMware support offerings can help meet your business needs, go to:

http://www.vmware.com/support/services

VMware Professional Services

VMware Education Services courses offer extensive hands-on labs, case study examples, and course materials designed to be used as on-the-job reference tools. Courses are available onsite, in the classroom, and live online. For onsite pilot programs and implementation best practices, VMware Consulting Services provides offerings to help you assess, plan, build, and manage your virtual environment. To access information about education classes, certification programs, and consulting services, go to http://www.vmware.com/services.

Introduction to VMware Workbench

<div style="text-align: right; font-size: 3em;">1</div>

VMware Workbench is an Eclipse-based development environment. The user interface retains the look and feel of Eclipse, but with VMware-specific extensions.

NOTE If you want to use the CLI to work with development kits, see "Command Line Usage" on page 14.

Installing VMware Workbench

The following sections cover prerequisites and how to install the VMware Workbench virtual appliance.

Hardware Requirements

In Workbench 3.5, the default memory for the Workbench virtual appliance is 3GB. To accommodate this default and maintain satisfactory performance, the physical machine that provides the virtual infrastructure for the Workbench virtual appliance should meet the following requirements:

- 64-bit dual core processor

- 4 GB RAM

- 40 GB of free HDD space

Download and Install VMware Workbench

In the VMware Developer Center, find the introduction page for the VMware Workbench VM. At the bottom of the page are a number of files for download.

- For vSphere, download the OVA file, an Open Virtualization Format (OVF) archive.

- For VMware Fusion, Workstation, or Player, download the ZIP file.

To run the Workbench virtual appliance on vSphere, place the OVA file on a network server or on your local machine, and use the vSphere Web Client to **Deploy OVF Template**. Answer questions in the dialog boxes. When finished, power on the Workbench virtual appliance.

To run the Workbench virtual appliance on VMware Fusion, Workstation, or Player, unzip the ZIP file into its own subdirectory of your Virtual Machines folder. In the VMware software, click **File > Open**. Navigate to the subdirectory, open the `*.vmx` file, and power on the Workbench virtual appliance.

In both cases, look at the console of the virtual appliance as it boots. Agree to the EULA by typing **yes** at the prompt. Optionally type **y** to have Eclipse auto-start. Press Enter to display the Linux login window.

VMware Workbench 3.5 is 64-bit

The physical host running your virtualization platform (VMware Workstation, VMware Fusion or ESXi) must support 64-bit code execution. Some servers include CPUs with support for Virtualization Technology (VT) but ship with VT disabled by default. If this is the case for your computer, you must enable it manually.

If you cannot boot VMware Workbench 3.5 after installation, try the following workaround:

To manually enable VT:

1 During the boot process, press **F12** to enter **System Setup**.

2 Select check box **Virtualization Support > Virtualization > Enable Intel Virtualization Technology**.

3 Select check box **Virtualization Support > VT for Direct I/O > Enable Intel VT for Direct I/O**.

4 Click **Apply** and click **Exit** to reboot.

Before You Report a Problem to VMware

When you encounter an error or problem with Workbench, a development kit, or a certification, and you want to report it to VMware, use the **Create Workbench Support Bundle** feature to create a ZIP file with much of the information support technicians need, including the following information:

■ The full Workbench system environment.

■ Information about all Eclipse plug-ins and features on your system.

■ All Workbench log files, and the Eclipse log file.

■ For Linux systems, some amount of `/proc` information.

Using this tool will expedite your support request because the technician will not have to ask you to gather the information before work can start on the issue.

To create the Workbench support bundle

To create the file while in the Workbench UI, perform the following steps:

1 In the Workbench Eclipse UI, click the **VMware** menu item and then **Create Workbench Support Bundle**.

2 In the VMware Support Bundle Creator pop-up window, accept the directory suggested for the file, or click **Browse** to select a different directory.

3 Enter a description of the error or problem, including how to recreate it (this is very important).

4 Click OK.

5 Attach this file to your DCPN case or initial email request for support.

Configuring VMware Workbench

Before you start work on certification or development, you may want to perform some of these set-up tasks:

■ Configure VMware Workbench to run remotely, or run it directly in the virtual appliance:

    ■ "Start VMware Workbench Remotely Using X Forwarding" on page 11.

    ■ "Start VMware Workbench Directly in the Virtual Appliance" on page 11.

■ Log in to Linux. See "Logging in to Linux" on page 12.

■ If you have a firewall, set HTTP and HTTPS proxies. See "Setting Network Proxies" on page 12.

■ If DHCP was selected, reconfigure the Workbench appliance to use a static IP address. See "Configuring Static IP for the Virtual Appliance" on page 13.

■ Install software plug-ins for your development kit or certification tests from the update site. See "Install New Software Instructions" on page 22.

■ Restart Workbench. After you install any or all of the software packages you need, click **Restart Now**. See "Installing New VMware Workbench Software Packages" on page 22.

Start VMware Workbench Remotely Using X Forwarding

You can set up Windows or Linux to display VMware Workstation remotely with X11 forwarding.

Using VMware Workbench Remotely from a Linux Desktop

To invoke the VMware Workbench remotely, open a terminal window on your Linux desktop and run the SSH command, enabling the X forwarding option. Root or superuser access is not required.

For example, use the following command after the shell prompt ($):

```
$ ssh -X -l vmware <vmwb-host> vmwb
```

For *vmwb-host*, substitute the host name or the IP address of the VMware Workbench virtual appliance. The default password for the `vmware` login is `vmware`.

Using VMware Workbench Remotely from a Windows Desktop

Use XMing and PuTTY to do X forwarding and open a remote client. You must start XMing first, then PuTTY.

NOTE Download and install XMing from the following location:

http://www.sourceforge.net/projects/xming/

Download and install PuTTY from the following location:

http://www.putty.org/

To start XMing and set additional parameters

1   From a Windows desktop, click **Start > Programs > XMing > XLaunch**.

2   Select **Multiple Windows** and click Next.

3   Select **Start no client** and click Next.

    The **A**dditional parameters dialog displays.

4   To enable cut and paste operations from Windows to the Workbench interface, type `-clipboard` in the **Additional parameters for XMing** text box.

5   (Recommended) Set the font size. Choosing 96 DPI for the font size dramatically improves readability for many users. To size the font in Workbench, type the DPI number in the **Additional parameters for XMing** text box, for example:

    `-dpi 96`

6   Click **Finish**.

To start PuTTY and log in to VMware Workbench

1   Start PuTTY and create a new session by typing the IP address of the Workbench virtual appliance.

    Save the session by typing a name in the **Saved Sessions** text box and clicking **Save**.

2   Select **X11** under **Connection > SSH**.

3   For Options controlling SSH X11 forwarding, select **Enable X11 forwarding** and click **Open**.

4   Log in as `vmware` with password `vmware`.

5   Type **vmwb** to start VMware Workbench.

Start VMware Workbench Directly in the Virtual Appliance

While installing the Workbench virtual appliance, you are queried about opening the UI automatically upon startup. If you reply yes, VMware Workbench will open when installation completes and whenever the virtual appliance starts. Otherwise you can start Workbench whenever you want.

You can log into the Linux virtual appliance and start Workbench from the command line.

Logging in to Linux

The VMware Workbench UI is delivered as a Linux virtual appliance. To log into Linux, you can use the default user account `vmware`, or `root`. The default password for both `vmware` and `root` is `vmware`.

NOTE   While Linux is booting, you will be asked to agree to the EULA. You must respond by typing **yes** for the boot process to continue. You will also be asked if you want to start VMware Workbench automatically after Linux reboots.

If the Workbench did not open, click the VMware Workbench icon on the virtual appliance desktop:

Or, launch a terminal in the virtual appliance (**Applications > Accessories > Terminal**) and type **vmwb**.

After you log in to Linux, create a workspace. You can also create a new user account after logging in.

Create a workspace

Create a workspace by typing in a workspace name, or accept the default presented to you.

Adding and Configuring a New User Account

You can add new users with the standard `adduser` utility, but check with your system administrator to ensure that users on this virtual appliance will interact correctly with users on other machines at your site.

Setting Network Proxies

The first time you start VMware Workbench, if you are behind a firewall you probably need to set proxies for HTTP and HTTPS, as well as proxy by-passes for local machines that you will connect to.

At first start up, a dialog displays that allows you to configure proxy settings.

In subsequent start ups, you can reach the proxy setting screen by selecting **Window > Preferences**. Expand the **General** preferences and click **Network Connections**.

To set HTTP proxies

1   Select **Manual** proxy configuration in the Active Provider drop-down menu.

2   Highlight **HTTP** and click **Edit**.

   Verify the proxy server name and port number are correct, or type the correct proxy and port number.

3   Select **HTTPS** and click **Edit**.

   Verify the proxy server name and port number are correct, or type the correct proxy and port number.

⚠ **CAUTION**   Do not edit the **SOCKS** (socket secure) proxy and authentication. Doing so may cause errors. Leave it unset.

---

4   In the **Proxy bypass** list, add systems on your local network (Intranet) that you want Eclipse to find. The following example shows various types of entries you might want to add:

```
10.*
*.example.com
127.0.0.1
localhost
```

   Wildcards can be used for IP addresses and domain names. Domain names can have only three parts.

5   Click **Apply** to save changes, and click **OK**.

   You are now ready to install development kits and certification tests.

Setting Proxies for the Linux System in the Virtual Appliance

The proxy setting in Eclipse is independent from the proxy settings for the Linux system. If you want to access the Internet with a browser outside of Workbench, use the following procedure.

To configure Linux for Internet access

1   Select **Computer** at the bottom of your virtual appliance console.

2   In the right pane of the Computer window, select **Control Center**.

3   In the left pane of the Control Center window, under Groups, select **System**.

4   In the System pane, select **Network Proxy**.

5   In the Network Proxy Preferences dialog, select the **Manual Proxy Configuration** radio button.

6   Select the **Use the same proxy for all protocols** check box.

7   In the **HTTP proxy** field, type the proxy value for your company, such as `proxy.examplecompany.com` and enter or use the arrow to change the **Port** value to the standard port for your company. You should see the values populated to all of the proxy boxes.

8   Click **Close** and close the Control Center window.

Click **Computer** again and choose an Internet browser, for example, Firefox.

Configuring Static IP for the Virtual Appliance

The VMware Workbench virtual appliance is initially configured for DHCP. From a terminal on the Linux console, you can change it to have a static IP address.

(Optional) Configure static IP

1   Access the virtual appliance (console view on Workstation, launch console in vSphere Web Client).

2   Log in as `root`, password `vmware`. If you log in as another user, you can `sudo` to run commands.

3   In the lower left corner, click **Computer > Control Center > Network Settings**.

4   In the Overview tab, select the primary network card and click **Edit**.

5   Select the radio button for **Statically assigned IP Address** instead of DHCP.

6   Type the IP Address, Subnet Mask, and Hostname that your network administrator assigned.

7   Click **Next**. Proceed to the Hostname/DNS tab.

8   Set your DNS (Name Servers) 1 and 2 and click OK. The network service will restart.

Accessing Help Resources

Kit-specific, Workbench-specific and general Eclipse help topics are available from the Help menu by selecting **Help Contents**. The Help browser displays a list of available help libraries. You can search all topics or narrow the scope using the search tool. Tabs at the bottom of the left pane allow you to view the following items:

■   Table of contents for any document

■   Consolidated index of all topics

■   Search results

■   Bookmarks you have set in the help topics

Navigating the Workbench UI with the Dashboard

To start working with the VMware Workbench UI, use the Dashboard. This feature is the first screen presented to you upon logging in for the first time. You can access this page from anywhere in the VMware Workbench UI by selecting **VMware > Dashboard** from the menu bar.

From the Dashboard, the following information and actions are available:

■ Information and instructions for common Getting Started tasks and problems

■ Download and install development kits

■ Download and install certification plug-ins

■ View a list of the plug-ins already installed

■ Download and install current debug symbols

■ Preview documentation before installing a plug-in

■ Check on the status of submitted certifications

■ Preview your certification entry in the VMware Compatibility Guide

■ View VMware partner news items

For instructions on how to use the Dashboard, see "The VMware Workbench Dashboard" on page 15.

Command Line Usage

If you prefer using the command line for editing, compiling, or debugging, use the Linux command line from inside the Workbench virtual appliance. The Linux command prompt has all the required toolchain and environment variables already set up for you.

Run package-extract from VMware Workbench

1 Download the development kit ZIP file to the appliance.

2 Run `package-extract`. For example:

```
$ package-extract -z <path> -d <dir>
```

The `<path>` is the file location of the ZIP file, and `<dir>` is the directory location where you want to output the development kit packages.

The VMware Workbench Dashboard **2**

The VMware Workbench Dashboard contains active links to assist your development kit and certification suite workflow. The Dashboard appears when you initially start VMware Workbench. Some links on the Dashboard are inactive or hidden until you sign in. If you dismiss the Dashboard, you can return to it later by selecting **VMware > Dashboard** from the menu bar.

This chapter covers the following Dashboard topics:

Signing In to See Content

Locate and click the blue **Sign In** button on upper right. Usually your sign-in ID is your company email address. You can choose whether to use your My VMware or Partner Central password. Some items are viewable only with Partner Central credentials.

After you sign in, your email address appears to the left of the **Sign In** button, and you can access content for all links on the left. Navigation bread crumbs will appear near the top, starting with **Home**.

Login Troubleshooting

If you fail to log in successfully, you will see one of the following error messages:

```
The account or password entered was incorrect.
Failed to connect through Dashboard.
```

Try the following remedies to fix the problem:

- Make sure you have configured network proxies correctly. See "Setting Network Proxies" on page 12 of this document.

- Make sure that the virtual appliance is connected to the network. For instance, verify that the Connected check box on the **VM > Settings > Network Adapter** page is selected.

- Verify that your VMware Workbench can access VMware Developer Center Repository by opening the web browser view (**Window > Show View > Other > General > Internal Web Browser**) and type the following URL for the VMware Developer Center Repository:

  ```
  https://vdc-repo.vmware.com/api
  ```

  You should see the date and time on the **vdc-repo** server. If this does not appear, ask your IT department to verify that you have access to the VMware Developer Center Repository site.

Workbench Panel

The top section on the left side provides information about VMware Workbench.

Getting Started

This line is selected initially, with helpful information displayed to the right. If you select another line, new information replaces the getting started instructions summarized below.

- First-time user? – Instructions for configuring the network proxy.
- Common issues and troubleshooting – Suggestions for solving common issues.
- Still have questions? – An email contact for getting questions answered.

News

VMware partner news and announcements about Workbench. Clicking a link opens a browser panel inside Workbench and displays the news or announcement.

Installed Packages

A list showing the VMware Test Manager and any certification suites or development kits already installed. Clicking a link leads you to start the Test Manager or to start a project for your development kit.

Certifications Panel

The middle section on the left side provides information about VMware certifications.

Certifications

Click the **Certifications** link to see a list of the certification kits available to you. If a message appears saying "you must log in to access available Certifications" then sign in with your company email address and your password for either My VMware or Partner Central.

After the list of Certifications appears, you can click on any certification kit that you are interested in. The Package Details pane appears with information about the certification, including a short description, links to install or download the package, a documentation link, the platform type, vSphere version, package version, and dependencies.

- You can preview documentation before installing a package by clicking the link to view it on-line.

- You can **download** a package to your local file system for installation on a machine not connected to the Internet. Specify download location. The file name is prefilled for you. For complete installation instructions, see "Installing Software Packages" on page 21.

- You can **install** a package to the machine you are working on. The installation wizard starts when you click to install the package. Update Site (Work with) is prefilled for you, so you need not cut and paste the Update Site from another location.

  During install the **Contact all update sites during install to find required software** check box is selected to pick up any dependent packages. Non-VMware update sites are disabled to avoid changing Workbench by installing incompatible software. If you want to install extra software, you must do it before installing any plug-ins. Click **Window > Preferences > VMware > Dashboard** and uncheck **Disable non-VMware Update Sites when installing VMware plug-ins.**

- From the Package Details pane, you can navigate back to any previous page by clicking breadcrumbs, or click **Certifications** to see the list of available packages again.

Compatibility Tools

There are three types of compatibility tools: (1) Certificaiton ID tools, (2) Server Certification Equivalency Tools (SCET), and (3) Compatibility Analysis Tools (CAT).

SCET and CAT are tools already available on Developer Center, but are now accessible through Workbench, after users sign in with their Partner Central credentials to see them.

Click the **Compatibility Tools** link to see a list of certification ID generators, and possibly other tools, to help with compatibility certification. After your organization joins a development program, you can use Cert ID tools to generate a certification ID for each device or module. You use this ID when submitting a certification. Cert ID tools are available for NetX, I/O devices, SRM, Server Storage, VASA, VVol vRO, vRealize, and others.

On-Demand Questionnaire Certifications

On-demand certification was added recently to the Dashboard. They are like traditional certifications but you fill out a questionnaire on the Web. To get started, click **On Demand Questionnaire Certifications**. A list of certification questionnaires will be downloaded. Click the questionnaire certification you want, and in the Package Details pane, **Click here** to access the certification. On the Questionnaire Certification pane, click **Create new request**. Answer questions on the Certification page. Click **Next** to reach subsequent pages.

Global Certification Waivers

If the computer you are using for certifications is not connected to the Web, not even through a proxy, see

Global Waivers for Internet Connected Computers

If the computer you are using for certifications is connected to the Web, click the **Global Certification Waivers** link to see a list of Product Types for which waivers are available.

Click **Download** to obtain the current list of Global Waivers. The global certification waivers list gets downloaded to /opt/vmware/waivers/waivers.xml by default, and soon an expandable list appears in the Global Certification Waivers > Product Types pane. You can refresh this list by downloading again.

In the Global Certification Waivers > Product Types pane, waivers are categorized: IO Devices, SRA (storage replication adapter), (ESXi) Server, Storage Array, VASA, and possibly others. Click one of the types to view the **Waiver Details** for that type.

The **Waiver Details** pane contains a list of the waiver IDs and a description of each. Clicking the waiver ID displays all available information about that waiver, including a short description, a detailed summary of the waiver, the list of waived tests, and the list of waived releases.

Some of the Product Type categories have a large number of waivers related to them. You can use the search box to restrict the list of waivers based on a keyword.

Figure 2-1. Searching waiver details

Waiver Details for Server 26 available

You can apply waivers to the tests you are running, or after tests have run, in the Test Manager by clicking the gearbox button ( ) then clicking **Apply Waivers** from the pop-up menu. Select the waivers you want to apply from the list and click **OK**. Only the waivers that apply to the certification will be applied.

Global Waivers for Non-Internet Connected Computers

If your certification computer is not connected to the Web, follow this procedure to install the waivers:

1 Install VMware Workbench Toolsuite on a computer that is connected to the Internet.
You can retrieve the Workbench Toolsuite from the VMware Developer Center:

    https://developercenter.vmware.com/group/workbench/toolsuite/3.5

2 Start Workbench Toolsuite and log in with your credentials.

3 On the Workbench Dashboard, under Certifications, click the **Global Certification Waivers** link to see a list of Product Types for which waivers are available. (The list of product types appears only if you have already downloaded the list of certification waivers.) Click **Download** to obtain the list of Global Waivers.

4   Specify a place on your hard drive for the `waivers.xml` file, for example `C:\temp`.
     Click **OK** and the `waivers.xml` file will be downloaded to the location you specified.

5   Copy the file to portable media and take the media to the non-Internet connected computer where your
     certification setup is run. On your certification computer where the VMware Workbench virtual appliance
     is installed, copy the `waivers.xml` file to the `/opt/vmware/waivers/` directory.

6   Start Workbench and open the certification session.

7   You can apply waivers to the tests you are running, or after tests have run, in the Test Manager by clicking
     the gearbox button (⚙) then clicking **Apply Waivers** from the pop-up menu. Select the waivers you
     want to apply from the list and click **OK**. Only the waivers that apply to the certification will be applied

Certification Submission and Status

To upload certification submission bundles, check the status of your submission, and view SR incident status,
click the **Certification Details and Submission Status** link.

Figure 2-2. Managing your certification submission
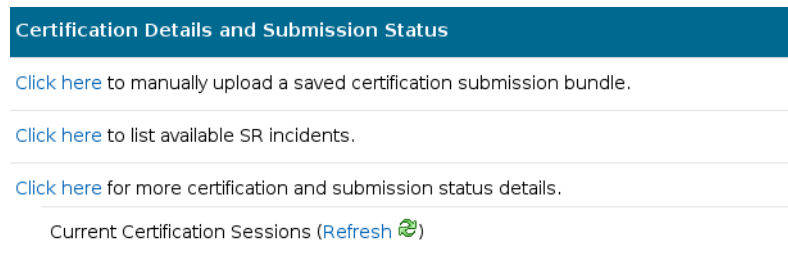
Manually Upload a Saved Submission Bundle

If you saved a submission bundle that you want to upload, click the top link **Click here to manually upload
a saved certification submission bundle**. In the resulting Upload Certification Submission Bundle dialog,
browse to locate your `*.zip` or `*.gzip` certification submission bundle, click **Upload**, and click **Finish**.

The resulting submissions upload properties will become visible in the Project Explorer view. If not already
created, a new folder called **manual_uploads** will be created that contains `Submission.prop` and `*.zip` or
`*.gzip` files in folders labeled by submission date, as shown in Figure 2-3.

Figure 2-3. Project Explorer Submission.prop file
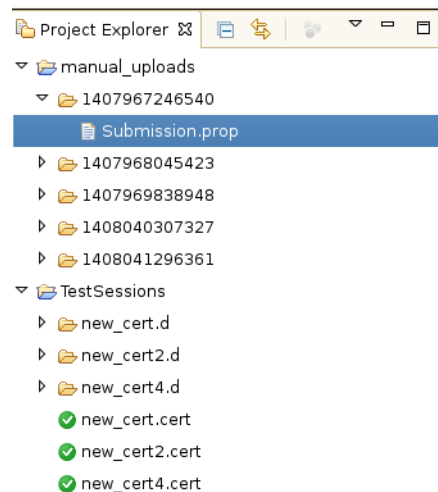
Click the `Submission.prop` file to open it in an editor for viewing. Further submission status can be accessed
in the Dashboard, as shown in Figure 2-4.

Available SR Incidents

To see all of the SR (service request) incident reports that you have filed, click the second link **Click here to list available SR incidents**. The resulting Certification Details and Submission Status pane contains your account information, a summary of SR incidents, and details of your SR incident reports.

Submission Status

All your current certification sessions are listed. To refresh the list, click the **Refresh** icon. To view the **Certification Submission and Status** details page, click **Click here**.

Certification Submission Status

To check status of your certification submission, click the third link **Click here for more certification and submission status details**. The resulting Certification Submission Status pane contains a list of your cert submissions with Cert Name, Cert Type, Transaction Id, Submission Status, and Cert Detail.

If you wish to export the submission status of your certifications in comma-separated variable (CSV) format, click the **Export** link. A dialog allows you to name the `.csv` file and choose where to store it.

If your certification submission status looks out of date, click the **Refresh** link and icon to update the status of your certification sessions.

Figure 2-4. Certification Submission and Status Detail Page

Once your submission has started the VMware review process, a clickable link appears under **Cert Details**. The Cert Details page contains product and certification details and lists compatible or equivalent products.

Add Transaction Ids

This section list submission status of certifications by a given transaction ID. If you know the transaction ID of a certification that was submitted on a different instance of VMware Workbench, you can add it by clicking **Add status by Transaction Id**, and keying in the ID. All columns, except the certification name, are populated with information about that transaction ID. You can delete the entry for an added transaction ID by clicking the **X** icon next to the entry.

VCG Preview

If you would like to see a preview of how your certifications will look in the VMware Compatibility Guide (VCG), you must have signed in using your VMware partner credentials. The page lists the ESXi versions you have certifications for, and product types for each version. Click the appropriate product type to see how your certifications will be listed in the VCG. If you find any errors in the entry for your product(s), notify VMware to have it corrected.

Development Panel

The Development section appears at bottom left of the Dashboard. You might need to scroll down to see it.

SDKs

Click the **SDKs** link to see a list of development kits available to you. If a message appears saying "you must log in to access available SDKs" then sign in with your company email address and either your My VMware or Partner Central password. After sign in, you can do the following tasks at this link:

- View the list of available uninstalled packages.

- Click the link for an available development kit to see Package Details.

  The Package Details pane provides information about a development kit, including short description, links to install or download the package, a link to documentation, platform, programming language, vSphere version, and dependencies.

  - You can preview the manual for a development kit before you install the package, by clicking the link to view on-line documentation.

  - You can **download** a package to a local file system for installation on a machine not connected to the Internet. Specify the download location. The file name is prefilled for you. For complete installation instructions, see "Installing Software Packages" on page 21.

  - You can **install** a package to the machine you are working on. The installation wizard starts when you click to install the package. Update Site is automatically filled in for you, saving you from having to cut and paste the Update Site from another location.

    > NOTE During install the **Contact all update sites during install to find required software** check box is selected to pick up any dependent packages. Non-VMware update sites are disabled to avoid changing Workbench by installing incompatible software. If you want to install extra software, you must do it before installing any plug-ins. Click **Window > Preferences > VMware > Dashboard** and uncheck **Disable non-VMware Update Sites when installing VMware plug-ins.**

- From the Package Details pane, you can navigate back to any previous page by clicking breadcrumbs, or click **SDKs** to see the list of available packages again.

Debug Symbols for VMkernel Updates

If you have not already done so, sign in to access the Debug Symbols repository. For debugging, you will need to update Workbench's debug symbols to match the version of ESXi host that is being tested.

To see a list of the available debug symbol files, click the **Debug Symbols for VMkernel Updates** link. To download a debug symbol file, click the link(s) of the build number(s) matching the ESXi build number for your host(s) under test, optionally change the save folder, and click **OK.**

Installing Software Packages

3

You can install software packages (plug-ins) from inside an Internet-connected Workbench virtual appliance, or download them to portable media (sneaker-net) and install them on a Workbench virtual appliance that is not connected to the Internet. One way to download development kits and certification suites is to use the Dashboard. Another way is Eclipse's Install New Software feature to download and install plug-ins.

This chapter contains the following topics:

- "Simplified Installation Instructions from the Dashboard" on page 21
- "Install New Software Instructions" on page 22
- "Installing Packages Offline (Sneaker-Net)" on page 24
- "Fixing Proxy Issues" on page 25
- "Uninstalling Software" on page 25

Simplified Installation Instructions from the Dashboard

The easiest way to install a software package in Workbench is from the Dashboard because the URL for the various SDKs and certifications is already filled in. The Dashboard is the first screen presented to you when Workbench starts up. However, you can always get to the Dashboard by navigating to **VMware > Dashboard.**

To install a software package from the Dashboard:

1 You must sign in to see the list of available SDKs or certifications. See "Signing In to See Content" on page 15.

2 On the left side, click either the **SDK** or **Certifications** link to see the list of SDKs or certifications that you can download and install.

3 Click the name of the SDK or certification that you wish to install.

4 Click the link to install the SDK or certification package to the local machine. The installation wizard starts after you click here to install. Update Site (Work with) is automatically filled in for you, saving you from having to cut and paste the Update Site from another location.

5 In the Install wizard, click **Select All**, and then click **Next**.

6 On the Install Details page, click **Next**.

7 Review the license, select **I accept the terms of the license agreement**, and click **Finish**.

8 During the install, a dialog box appears telling you the module is unsigned. Click **OK.**

9 At the end of the installation, a dialog box asks you for permission to restart VMware Workbench. If you are done installing software packages, click **Yes**. If you have more to install, click **No**.

Install New Software Instructions

You can install software using the classic Eclipse method where you provide the URL for a software package. This allows you to specify arbitrary software packages for a custom approach.

Specifying Available Software Sites

There may be multiple software sites from which you must download packages. Sometimes a package on one site has a dependency on a package on another download site. VMware Workbench can coordinate this for you and automatically download the components you need to satisfy any dependencies. To set up this feature, use the **Available Software Sites** preferences.

Set up Available Software Sites preferences

1   From the VMware Workbench click **Help > Install New Software** and click the **Install** link for the plug-in.

2   From the Install dialog box, click **Available Software Sites**.

The Preferences dialog box displays with **Install/Update** expanded and **Available Software Sites** selected.

3   Click **Add** to display the Add Repository dialog box.

4   Determine the URL you need from your partner site on the Developer Center or from another source.

Enter, or copy and paste, the URL of the download site in the Location text box.

5   Type a name for that update site location in the **Name** text box. Click **OK**.

The update site appears, added to the list of available software sites in the Preferences dialog box.

6   The status of the site should be Enabled. If the status of the site is Disabled, select the site and click **Enable**.

7   Repeat the **Add** operation until you have added and enabled all the available software sites that you will be using for VMware Workbench.

8   When you are finished, click **OK** to dismiss the Preferences dialog box and return to the Install dialog box, where you can proceed to installing new software packages.

Installing New VMware Workbench Software Packages

You can add VMware Workbench software packages and certification tests or update existing packages using the Install dialog. You can reach the Install dialog two ways: by clicking **Help > Install New Software,** or by clicking **Install** in the Dashboard Development panel under SDKs, after you have logged in.

Software packages can be installed from a site you already entered using the Available Software Sites dialog, from a network update site, or from a local folder (after downloading the ZIP files to a local folder).

IMPORTANT   Before you install any software packages, proxy settings must be configured. Failure to configure proxy settings can result in the installation stalling. You were offered the opportunity to set proxies when you installed the base VMware Workbench appliance. If you have not already configured the proxy settings, see "Fixing Proxy Issues" on page 25.

Install software packages from the list of available software sites

1   From the Install wizard, make a selection from the **Work with** drop-down menu.

2   Click **Select All**, or select the check box next to the package name, and click **Next**.

The system calculates the requirements and dependencies for the package(s) you selected.

NOTE   The **Contact all update sites during install to find required software** check box is not selected by default here in the Install wizard, unlike when installing plug-ins from the Dashboard. Additionally, non-VMware Update Sites are disabled here as they are when installing from the Dashboard.

(Optional) You can expand **VMware Workbench <Package Name>** in the Name column to see the list of components included in the package. You can choose to exclude some or all components by deselecting them, however this step is not necessary because the installation software prevents you from installing duplicate or unnecessary components.

3 Review the **Install Details**, and click **Next**.

4 Review the license text, select **I accept the terms of the license agreements**. You cannot install the software without agreeing to the license terms. Click **Finish**. The software installation can take several minutes to complete. If you are asked to accept unsigned content, click **OK**.

5 When installation is complete, a dialog box advises you to restart Eclipse for the changes to take effect and asks you if you want to restart now.

If you have other packages to install, click **No**.
If you are finished installing software packages, click **Yes** to restart Eclipse.

Install software packages from the network update site

1 Open a Web browser and go to the VMware download site for your development kit or certification suite.

2 Copy the URL for the software package to download.

3 Click **Help > Install New Software**. From the Install dialog box, click **Add**.

4 The first time you install the package, paste the URL into the Location text box. Type a Name. Click **OK**. **VMware Workbench <Package Name>** shows in the text box.

If you are reinstalling the package (for an update of the software), select the update site from the list in the **Work with** drop-down menu.

5 Select the check box next to the package name, and click **Next**.

The system calculates the requirements and dependencies for the packages you have selected.

(Optional) You can expand **VMware Workbench <Package Name>** in the Name column to see the list of components included in the package. You can choose to exclude some or all components by deselecting them, however this step is not necessary because the installation software prevents you from installing duplicate or unnecessary components.

6 Review the Install Details and click **Next**.

7 Review the license text, select **I accept the terms of the license agreements**. You cannot install the software without agreeing to the license terms. Click **Finish**. The software installation can take several minutes to complete. If you are asked to accept unsigned content, click **OK**.

8 When installation is complete, a dialog box advises you to restart Eclipse for the changes to take effect and asks you if you want to restart now.

If you have other packages to install, click **No**.
If you are finished installing software packages, click **Yes** to restart Eclipse.

Install software packages from a local folder

1 In the Dashboard, you can choose to download a package to a local folder. Alternately, you can open a browser in the Workbench virtual appliance and go to a VMware download site. Download the ZIP file containing a software package. For example, download the following file:

```
UpdateSite–<package name>–com.vmware.vide–<version number>.zip
```

2 Make a new directory and move the ZIP file into it, for example /home/vmware/update.

3 In VMware Workbench, click **Help > Install New Software**.

4 Click **Add** to open the Add Repository dialog box.

5 Click **Local**, navigate to the update file path (repository root directory) and click **OK**.

6 Back in the Add Repository dialog box, type a name for the package and click **OK**.

7   Click **Select All**, or select the check box next to the package name, and click **Next**.

The system calculates the requirements and dependencies for the package(s) you selected.

(Optional) You can expand **VMware Workbench <Package Name>** in the Name column to see the list of components included in the package. You can choose to exclude some or all components by deselecting them, however this step is not necessary because the installation software prevents you from installing duplicate or unnecessary components.

8   Review the **Install Details**, and click **Next**.

9   Review the license text, select **I accept the terms of the license agreements**. You cannot install the software without agreeing to the license terms. Click **Finish**. The software installation can take several minutes to complete. If you are asked to accept unsigned content, click **OK**.

10 When installation is complete, a dialog box advises you to restart Eclipse for the changes to take effect and asks you if you want to restart now.

If you have other packages to install, click **No**.
If you are finished installing software packages, click **Yes** to restart Eclipse.

Installing Packages Offline (Sneaker-Net)

To accommodate those who do not have Internet access for installation of the software packages, VMware Workbench installation packages can be downloaded and transferred (sneaker-net) as individual ZIP files. You can download software packages from the same site where the base VMware Workbench package resides.

Install software packages off-line

1   On a machine connected to the Internet, using the Dashboard of the Workbench virtual appliance or Workbench IS, click the "download" link for one of the SDKs or Certifications. Download the software package as a ZIP file. Copy it to portable media. For example, download and copy the following file:

    UpdateSite–<package name>–com.vmware.vide–<version number>.zip

2   Insert the portable media into the system running the Workbench virtual appliance where you wish to install the package.

3   On the Workbench virtual appliance, make a new directory, for example /home/vmware/update, and copy the ZIP file from portable media into the update directory.

4   In VMware Workbench, click **Help > Install New Software**.

5   Click **Add** to open the Add Repository dialog box.

6   Click **Local**, navigate to the update file path (repository root directory) and click **OK**.

7   Back in the Add Repository dialog box, type a name for the package and click **OK**.

8   Click **Select All**, or select the check box next to the package name, and click **Next**.

The system calculates the requirements and dependencies for the package(s) you selected.

(Optional) You can expand **VMware Workbench <Package Name>** in the Name column to see the list of components included in the package. You can choose to exclude some or all components by deselecting them, however this step is not necessary because the installation software prevents you from installing duplicate or unnecessary components.

9   Review the **Install Details**, and click **Next**. Review the license text, select **I accept the terms of the license agreements**. You cannot install software without agreeing to the license terms. Click **Finish**. Installation can take several minutes to complete. If you are asked to accept unsigned content, click **OK**.

10 When installation is complete, a dialog box advises you to restart Eclipse for the changes to take effect and asks you if you want to restart now.

If you have other packages to install, click **No**.
If you are finished installing software packages, click **Yes** to restart Eclipse.

Fixing Proxy Issues

If you have problems with the installation seeming to stall or taking a long time to complete, or you cannot open a terminal, you might need to configure proxy settings.

Fix proxy issues

1   In VMware Workbench (Eclipse) menu bar, click **Window > Preferences**.

2   On the left side, expand **General** and click **Network Connections**.

3   In the Active Provider drop-down menu, select **Manual** proxy configuration.

4   Verify that you have the HTTP and HTTPS proxies set properly. If not, select them one at a time, click **Edit**, and type the correct settings for the proxy host and port.

    a   Verify the proxy server name and port number are correct.

    b   In the **Proxy bypass** list, add the list of IP addresses or host names for which the proxy server should not be used. Use of wildcards is permitted. The list might look something like this:

```
*.example.com
127.0.0.1
localhost
```

5   Click **Apply** to save changes.

6   Click **OK** to exit.

Uninstalling Software

Also see the Eclipse *Workbench User Guide* for detailed instructions about removing software packages that are already installed.

> **CAUTION**   Even though it is possible to remove software packages, VMware advises against it. If you must do it, VMware urges you to be extremely careful because you risk rendering other packages inoperable because of missing dependencies. Data loss is possible.

Many installed packages contain common modules that are needed by other installed software packages (development kits and certification suites). If you inadvertently remove a common module that other installed software depends on, you can render those software packages inoperable.

If you have used the software to create valuable data that you want to keep, take measures to copy the files to a safe location before uninstalling the software.

Uninstall software

1   In the VMware Workbench (Eclipse) menu bar, click **Help > About VMware Workbench**.

2   Click **Installation Details** to see the Installation Details page.

3   On the **Installed Software** tab, select the package to uninstall, and click **Uninstall**.

Running Test Manager

<div style="text-align: right">**4**</div>

VMware Workbench provides a Test Manager to help you run certification tests or unit tests. In unit test mode, you can run the certification tests individually and change the test parameters.

For information about your specific certification, see the certification guide appropriate to your module type. You can fine PDF certification guides at the VMware partner site, or in Eclipse, **Help > Help Contents**. This chapter contains the following Test Manager topics, generally in workflow order. When you want to complete a certification from start to finish, you should be reading the certification guide, not this chapter.

To set up the appropriate testbed for your certification, run the **Installation Helper** before starting the Test Manager. For information on the Installation Helper, see the *VMware Workbench Installation Helper Guide.*

When running in unit test mode, you still see references in the UI to certifying the results. Unit test results cannot be use for certification; the results are only for your use in testing.

Sections "Test Manager Command-Line Interface" on page 38 and "Test Manager Web Client" on page 40 describe alternate interfaces to Test Manager. You might find them useful.

You can record your certification session for later replay from the command line. This allows you to run the session remotely and unattended. For more information, see "Test Progress page" on page 32, and "Running Multiple Recorded Test Sessions Simultaneously" on page 38.

Many certifications require that you fill out a configuration information page. You must enter this information each time you run the certification. Test Manager saves the parameter values in a parameter cache and can prefill them for you when you run the certification again. You can import the cache from other workspaces. During your Test Manager session, you can override the prefilled information to suit your individual needs for this certification run. For instructions on configuring this feature, see "Enabling Prefilling from the Parameter Cache" on page 41 and "Importing a Parameter Cache from Another Workspace" on page 41.

Tests Dashboard

The Tests Dashboard is a utility plug-in installed with Test Manager on your Workbench system. The dashboard displays in a view separate from the Test Manager on the Workbench interface. Use the Tests Dashboard to view the state of all test sessions in the Test Sessions folder. The sessions display even if they are not currently running.

Viewing the Test Manager Dashboard

Open the Test Dashboard view from the Window menu. (**Window > Show View > Other**, then expand **VMware** and select **Test Dashboard**)

The following table lists the various dashboard view column headings and a description of the data you see in each column.

Table 4-1. Test Manager Dashboard View Contents

Column Heading	Description
Certification	The name of the certification session.
Total tests	The total number of tests in this certification.
Passed/Waived/NA	The number of tests that passed, were waived in the last run, or were not applicable.
Failed	The number of tests that failed in the last run.
Remaining	The number of tests that did not run in the last run.
Status	Status of the certification session: `Closed`, `Not Running`, `Running`, `Waiting`, `Run Ended`.
Progress	A moving colored bar that tracks progress of the certification tests. Green indicates that tests have passed. Red indicates that tests have failed.

Scheduling Test Sessions Using the Test Manager Dashboard

On the dashboard, a **Schedule** link shows to right of the **Progress** column for test sessions that are open.

Scheduler limitations

1   The test session must be open to schedule it.

    If you want to schedule a test session and don't see the **Schedule** link next to it, open the test session by double-clicking the session's `.cert` file name in the Navigator pane.

2   Some test sessions require the user to enter configuration data in one of the test session pages. You must make sure data has been entered and the **Confirm** button has been clicked before you schedule the test.

3   Some certifications ask the user for information at run time. A test that is scheduled but requires runtime input will not automatically run to completion. The test will wait for user input until the requested information is entered.

4   Only the tests that are checked in the Test Selection page will be run during the scheduled test.

5   Test sessions that are designed to be run manually cannot be scheduled.

Schedule a Test Session

1   Click the **Schedule** link.

     The Schedule a Test Session dialog box displays.

2   Select a starting date on the calendar.

3   Adjust the time field.

     a   Highlight the hour, minute or second number.

     b   Click the top button next to the time field to adjust the number higher. Click the bottom button to adjust the number lower.

4   Select the Frequency to run the test from the drop-down menu.

     Choices are **Once**, **Hourly**, **Daily**, or **Weekly**.

5   Click **Schedule**.

Cancel or Reschedule a Test Session

A scheduled test session can be cancelled (unscheduled) or rescheduled by clicking the **Schedule** link next to the Progress column for the test.

In the Schedule a Test Session dialog box, to reschedule a test session, reset the date, time or frequency of the test and click **Schedule**.

To cancel a test session, click **Unschedule**.

Viewing the Test Session Web Report

You can view a read-only summary report of each test session with its test results on a web browser by typing the following URL in a browser: `http://`*IP address of Workbench virtual appliance*`:8100/tm`.

You can retrieve the IP address of the virtual appliance by running `ifconfig` from a terminal window inside the VMware Workbench virtual appliance.

The content of the web report is the same as the Tests Dashboard view in the Workbench interface, except that there are no Schedule links in the web report.

Starting a New Test Session

You can create a new test session or start an existing session. You can optionally use a cheat sheet to get started.

Use a cheat sheet to get started

1   Select **Cheat Sheets** from the Help menu.

     Help > Cheat Sheets

2   From the Cheat Sheet Selection dialog box, expand **VMware Test Manager**.

3   Select **Creating a Test Session** and click **OK**.

     Follow the instructions for creating a test session.

Open Test Manager and create a new test session

1   Open Test Manager.

From the Window menu, select **Open Perspective > VMware Test Manager**.

2   Create a new test session.

From the File menu, select **New**, then select **Test Session**. (**File > New > Test Session**)

The **New Test Session** wizard displays.

To import an existing test session, see

3   In the **New Test Session** wizard, complete the various fields:

Table 4-2. New Certification Session Fields

Field Name	Description
Test Project	Click the Browse button to choose a project for this test. The **Folder Selection** pop-up window displays a list of the project folders and the TestSessions folder. If you choose a project, the test session file will be saved to the project folder. If you choose TestSessions, the test session file will be saved to that folder.
Session Name	Name the certification session so you can distinguish it from other tests and your real certification sessions. The default name is new–cert.cert.
Categories	Choose a category from the drop-down menu. Your choice determines which tests are preloaded for your use. The list of available session types loads in the **Session Types** text box.
Session Types	Select the **Session Type** to test.

4   (Optional) For unit testing, select the **Unit Test Mode** check box.

In unit test mode, you can modify the test cases and specify parameters. You cannot do a full certification session from this mode. You can submit troubleshooting logs from a unit test. If you select this mode, **[Unit Test Mode]** shows at the top-right corner of the subsequent Test Manager screens.

5   Click **Finish**.

The certification session starts. The screens vary somewhat according to the type of testing you are doing.

See the appropriate certification guide for detailed information on the certification you are going to run.

Setting Up and Running the New Test Session

You configure the test session by entering testbed information, server configuration, and selecting tests to run.

Before you run Test Manager, you should have already run Installation Helper to set up your testbed and create the virtual machine library needed for device driver testing. See the *VMware Workbench Installation Helper Guide*, found in the Help Contents.

Before running your certification, you can configure Test Manager to save any configuration page information you enter, or you can import configuration information from another session that you ran previously. For configuration instructions, see

Host Selection page

1   If you have downloaded Global Waivers, apply them now by clicking the gear icon found on the right side of the bar that contains the **Back** and **Next** buttons.

a   The Waiver Selection window lists the waivers that you downloaded in the Dashboard.

b   Select the waivers you wish to apply to the tests you are about to run and click **OK**.

c   A pop-up message tells you the waiver was applied successfully. Click **OK** to dismiss the window.

After the tests have run, you can verify that the proper tests were waived on the Test Selection or Summary page.

    d   You can apply waivers after a test run in the Summary page by clicking the **Apply Waiver** link.

2   Complete the **Primary Host** and if required, **Secondary Host** information.

ESXi hosts must have SSH enabled. If you have existing connections to ESXi hosts defined in VMware Workbench, choose them from the drop-down menus. If not, you can establish a new connection by clicking Add Target to bring up the New Connection dialog box. If you add a new host connection, set the proxy from the Window menu.(**Window > Preferences > General**)

3   Click **Check Host** for the Primary Host and for the Secondary Host, if present.

For the primary host only, the **Check Host** feature checks VIB dependencies for IOVP, Storage VAAI NAS, and PSA certifications. It also checks VIB dependencies for CIM Provider Acceptance Tests and Host Profile Acceptance Tests.

NOTE   You must perform the **Check Host** check for the Primary Host. The **Next** button is dimmed until you click **Check Host** for the Primary Host.

After the VIB dependencies are checked, you see either **Passed** or **Failed** in the **VIB Dependencies** field.

If the VIB dependency test fails, you see a VIB Dependency Error dialog that explains its findings and suggests actions to take. In general, you can try the following steps if the VIB dependency fails:

■   Uninstall redundant VIB packages. The system performing certification tests should be clean machine, which has only the VIB packages required to run your certification tests. Removing unnecessary VIB packages will reduce the chance of failure.

■   Check for missing dependencies between your VIBs. If you are sure your VIB packages don't have any dependencies with each other, you can ignore this message and go to the next step. False warnings may happen when you install VIB packages of different classes. For example, if you install both a 3rd party network driver and a storage driver, it may give a false warning.

4   (Optional) If available, click **Check Testbed** to use the Testbed Analyzer tool to verify your ESXi systems against the configuration requirements of your certification.

IMPORTANT   The Testbed Analyzer is a separately installable plug-in. If the plug-in is installed and the open certification supports testbed analysis, the Check Testbed button is present.

The Testbed Analyzer view displays the results of the analysis. You can view the results in Tree mode or Text mode. Switch between modes by clicking the tabs at the bottom of the view.

Errors are flagged with red icons. You can expand the directories for each error to inspect the error. In Tree mode, the first error is automatically expanded.

Fix any problems with your testbed configuration before proceeding with the certification

5   Click **Next**.

Server Configuration page

Some certifications require you to make selections concerning the testbed server. You will not see this page if it does not apply to your certification.

1   Select the check boxes that apply to your test session.

2   Select either **Partial Certification** or **Complete Certification**.

3   (Optional) If you choose **Partial Certification**, select the components have been upgraded since the last certification.

4   Click **Confirm**.

Once you have clicked Confirm, you cannot go back and change the selections.

5   Click **Next**.

Questionnaire page

Before you can continue to test selection, certain types of certifications require you to fill out a questionnaire. If the questionnaire page is present, select the answers to the questions and click **Confirm**.

If you plan to run the tests repeatedly and don't want to repeatedly fill out the questionnaire page, you can save the answers to a workspace file.

Test selection page

1   From the Test Catalog, select **All tests** or expand it to reveal groups of tests you can select or expand. You can keep expanding the sections to get down to each individual test and can select or deselect individual tests.

2   (Optional) You can select specific tests and apply a waiver to them by right clicking on the test.

    For more information about applying waivers, see "Entering or Modifying a VMware Waiver Approval ID" on page 37.

3   (Optional) You can enter or modify comments on specific tests by right-clicking on the test.

    The Add a Comment pop-up window displays where you can type a comment. Click **OK** when you are finished typing.

4   Once you have selected all the tests for this run, click **Next**.

Configuration page

1   If presented to you, fill in the required configuration information.

    Most of the Storage Certifications require that you enter information in a **Configuration Information** form before any tests can be run. Verify or change the various configuration parameters used for your testbed.

    NOTE   You can save your inputs so that subsequent certification runs can reuse the configuration information you enter for this page. Or you can import configuration information that you entered in another workspace. For instructions, see "Enabling Prefilling from the Parameter Cache" on page 41 and "Importing a Parameter Cache from Another Workspace" on page 41.

    ---

2   (Optional) A toggle button at the bottom of the form allows you to hide or view virtual machine information for each field. (**Show VM Info/Hide VM Info**)

3   Click **Confirm** and click **Next**.

For more details on some of the data fields in the Configuration form, see the Storage certification guide.

Test Progress page

1   If you want to record the test session for future replay from the command-line, select the **Record Test Session** check box.

    IMPORTANT   The record function doesn't work with manual tests.

    For more information about the Record Test Session feature, see "Running Multiple Recorded Test Sessions Simultaneously" on page 38.

2   Click **Run Test Session**, or optionally if present, click **Run Unattended**.

    The page displays the tests you have chosen in one text box and the results in another text box.

    IMPORTANT   **Run Unattended**: If this mode is available for your certification, and you will not have to perform some action, such as removing a cable or powering off some equipment before continuing the test, then you can click **Run Unattended** instead of **Run Test Session**.

    If you are rerunning a certification and entered test data for the previous run, that test data was recorded. If you select **Run Unattended** for this run, the data will be reused and you will not have to reenter it.

When the test finishes, the results indicate if the test passed or failed.

IMPORTANT   The tests to be run are signed by VMware. A test-checking program determines if any of the tests have been altered. If any altered tests are found, an Error dialog box opens with the altered tests listed. Results of the these tests will be invalid for official certification. Click **OK** to continue the test session.

---

3   (Optional) Click **Abort Test Case** or **Abort Test Session** to stop a test or the entire test session before it finishes.

4   (Optional) Click **Rerun Test Session** to rerun the entire session.

5   Click **Next**.

Session Summary page

1   View the test Summary report and Test Run Details report.

If there were problems that caused the test session to fail and quit, the Test Run Details report shows you which tests were skipped. Fix the problems that caused the failure and rerun the test session.

NOTE   Clicking the **Help** button brings up this *Getting Started Guide*.

You can apply Global Waivers for tests that did not pass by clicking the **Apply Waivers** link.

2   (Optional) If you are satisfied with this run, you can create a submission report and package by clicking **Upload/Save Results**.

The Results Submission form displays. For instructions on how to fill out this form and generate the report, see "Generating a Results Package" on page 34.

3   (Optional) If you plan to submit the results package from this test machine, click **Purchase SR** to be taken to the VMware Store page (http://www.vmware.com/go/enablement) where you can purchase the SKU package. that allows you to submit your certification.

The CLI application locks the workspace when performing any operation. This has the following implications:

■   You can run a separate instance of the CLI application, but you must use another workspace.

■   The CLI application and the Workbench UI can share the same workspace, but they can not run at the same time.

Running an Existing Test Session

To rerun an existing test session, perform the following steps:

1   Depending on what you were doing in Workbench up to this point, the contents of the left pane can vary. If neither the Project Explorer nor the Navigator tab is open, open the Test Manager. (**Window > Open Perspective > VMware Test Manager**)

2   In either the Project Explorer or the Navigator tab in the left pane, locate the name of the test session you wish to open.

Test sessions are files that have the `.cert` suffix. They can be located either inside your project (expand the project to find it) or under TestSessions (expand TestSessions).

3   Double-click the name of the test session you want to open.

4   Run the test session.

NOTE   If the test was in Unit Test Mode, you can make changes to it, if not, you must run the test as is.

When the tests have finished and you are ready to generate a results report. For instructions on how to generate a results report, see "Generating a Results Package" on page 34.

Generating a Results Package

After you have run the tests and passed all of the tests, or added waivers for the ones that didn't pass, and you have tested deployment of your package with VUM, you are ready to submit a results package to VMware.

NOTE   For a statement of best practices before submitting your VIB, see "Verifying Remote Installation Before You Submit Your VIB to VMware" on page 44."

Before You Start

Before you generate a results package, you should consider removing extraneous test result directories so that the ZIP file created for the submission is not too large. Currently the upper limit on ZIP size is 1 GB. If you have performed multiple runs of the tests, you will have a separate timestamped directory for each test run. You only need to submit the test run that passed; the other test runs will be ignored anyway, so removing them will not affect the final result. To decrease the size of the ZIP file, delete the extraneous timestamped directories, or move them to another location.

Generate the Results Report

IMPORTANT   You must have configured your proxy settings in order to upload your results to VMware. See "Setting Network Proxies" on page 12.

1   In the **Results Submission** pop-up window, enter your VMware certification ID in the **Cert ID** field.

    If you have not received one, please contact VMware.

2   (Optional) Click **Troubleshooting submission** if you are uploading troubleshooting logs to VMware.

3   (Optional) Enter up to 1000 characters in the Comment field.

    The content of the comment field is added to the certification submission confirmation email, and is stored in the `cert_<cert ID>.pdf` as the last field in the Partner and Test Lab Information table. The comment also is stored in the `report.xml` file.

4   Click **Next**.

5   Enter organizational and vendor information as requested in the Result Submission form.

    The Partner Information fields are pre-filled if you have previously submitted results.

6   (Optional) Click **Preview Report** to view the report before sending.

    The report can take a few minutes to complete. You do not have to wait for the report to complete before continuing on with other tasks.

7   Click **Finish** to start gathering information for the submission.

    A pop-up window tells you how long this step will take.

8   Click **OK** to start results collection or click **Cancel** to return to the Summary page and upload test results at a later time.

9   (Optional) For some certifications, the **Certification Verifier** wizard displays.

    Fill in the fields with the required information. All of the fields must be filled with the appropriate information in order for your certification submission package to be complete.

    The information required is specific to each certification type. See your VMware certification program guide for details. If you have questions or have difficulty obtaining the necessary information, contact your VMware program representative.

    If you cancel this page, the process continues, but a warning informs you that your submission might not be accepted without the required information. If you finish the form, but leave some of the fields blank, the process continues, but you might be required to supply the information later, which could delay your submission.

10   Upload the report to VMware, or save it to a file.

When the submission report completes, a pop-up window displays.

- If you have an Internet connection on the test machine, you can click **Yes** to submit your certification results now. To continue the submit process, see "Submitting the Test Results Immediately" on page 35.

- If you do not have an Internet connection or you just wish to delay submitting your results, click **No** to store the file.

  A dialog displays the location of the submission file. Note the location.

- If you have to move your submission package to a computer with Internet access, copy the bundle to a removable media device.

  For instructions on how to submit your results at a later time or on a different machine, see "Submitting Results at a Later Time" on page 36.

Submitting the Test Results Immediately

Use these instructions if you chose to submit your test results from the test machine immediately after running the tests. After you clicked **Yes**, the report starts uploading to VMware.

1   When the upload completes, a pop-up window displays the transaction ID. Note the transaction ID for future reference.

    The transaction ID is stored in the `submissions.log` file in the `cert.d` directory, where `cert` is the name you gave your certification session. The entry lists the timestamp and the transaction ID.

2   When the upload is complete, a **Create new Service Request** (SR) pop-up window displays that allows you to automatically file an SR. Click **Yes** to continue.

3   Complete the Certification Submission wizard.

    For instructions, see "Filing a Service Request (SR) with the Certification Submission Wizard" on page 35.

Filing a Service Request (SR) with the Certification Submission Wizard

After uploading results, if you clicked **Yes** to file an SR, or you clicked **File SR** in the Test Manager Summary page, the **Certification Submission** wizard displays.

NOTE   You can also open the **Certification Submission** wizard from the New Wizard dialog from the right-click menu found in the Navigator pane (**New > Other > VMware > SR Submission Wizard,** then click **Next**).

---

1   In the **VMware Service Request Login** page, login to have your credentials validated.

    Fill in your `vmware.com` username and password and click **Login**.

2   If you have not already previously submitted the transaction ID, a pop-up window displays asking for you to enter the transaction ID for your submission.

    The system displays your partner information, including how many available credits you have.

    If you have enough credits to submit an SR, the next screen is enabled. Otherwise you must go to the VMware Store to purchase enough credits, and then come back to this wizard to file the SR (click **File SR**).

3   In the **Service Request Information** page, fill out all the required information.

    The system auto-fills all the known information. You must enter some specifics for this certification.

    a   Select the **Account** from the drop-down list.

    b   Select the **Product Name** from the drop-down list.

    c   Select the **Incident pack**age to use from the drop-down menu.

    d   Fill in the phone number, country and time zone if not auto-filled.

e   Fill in the optional information fields if desired, such as secondary email address, any alternate contact information, or a third party tracking number.

f   Select the **Problem category** from the drop-down menu or fill it in if the drop-down menu is blank.

For example, you might choose **Server**, or **Storage**.

g   The **Description/Transaction ID** field is auto-filled if possible. If it is not, enter the transaction ID in the text area.

4   Click **Finish** to complete the certification submission process.

The system automatically files the SR.

You can check the status of your certification submissions in the Dashboard. For an explanation of the Dashboard, see Chapter 2, "The VMware Workbench Dashboard," on page 15.

Submitting Results at a Later Time

If you choose to submit your results at a later time, save your submission results and the location and transaction ID that are supplied to you. When you want to submit the results, do the following, or if your test session was run on a computer not connected to the Internet, you can first move the ZIP file to a flash drive or other removable media and move it to a computer that is connected to the Internet and has VMware Workbench virtual appliance or Toolsuite installed and then do the following:

1   Start the Test Manager. Navigate to **Window > Open Perspective > VMW Test Manager**

2   Start the **Certification Submission** wizard. Right-click in the Navigator pane and select **New > Other**

3   Expand **VMware** in the Wizards list.

4   Select **SR Submission Wizard** and click **Next**.

5   Browse to the directory with the submission file.

The submission package is stored as a ZIP file in the `cert.d` directory, where `cert` is the name you gave your certification session.

6   Upload the submission package.

When the upload is complete, you will see the **Submission Status** filled in, including the transaction ID. You should record the transaction ID for future reference. Click **Next** to proceed to submit the SR. For instructions, see "Filing a Service Request (SR) with the Certification Submission Wizard" on page 35).

Off-Line Submission for Certification

The easiest way to submit a certification bundle to VMware is to click the **Upload/Save Results** button on the Summary page displayed after you finish running a test session. However, if you are working in a lab that is not Internet connected, it is not possible, so you must do an "off line" submission of the certification bundle.

To submit a certification bundle off-line

1   Save the certification bundle by clicking the **Upload/Save Results** button. Fill out the next three dialogs normally, but when asked "Do you want to submit your results automatically?" click **No**. The submission bundle will be saved as a ZIP file, and you will see its location with a summary of how to submit it.

2   Copy the ZIP file to an Internet connected machine where you installed either Workbench or Workbench Toolsuite. The ZIP can be large – 100MB files are common, and several GB files are sometimes generated.

3   Upload the ZIP file to VMware. Using either Workbench or Workbench Toolsuite, in the Dashboard select **Certification Details and Submission Status**. Click the first link, for manually submitting a certification. In the next dialog enter the location of the ZIP file on a local file system.

If the last step does not work, you can use a browser to visit the following URL and upload the ZIP file: http://partnerweb.vmware.com/cva_results/cert_result_upload.php

Entering or Modifying a VMware Waiver Approval ID

During test selection, you have the option of waiving any tests that are not pre-certification or post-certification tests.

1   From the Test Catalog, right-click on the test you want to waive.

2   Select **Enter/Modify VMware Waiver Approval ID**.

      When the Test Waiver dialog box displays, type the waiver ID supplied by your VMware representative and click **OK**.

3   (Optional) You can remove a waiver by selecting **Remove VMware Waiver Approval ID** from the right-click menu.

      The test can now be selected for the test run.

4   (Optional) You can enter a comment by selecting **Enter/Modify A Comment** from the right-click menu.

      Type a comment in the Add a Comment dialog box that displays. Click **OK** to add the waiver comment.

5   (Optional) You can remove a comment by selecting **Remove a Comment** in the right-click menu.

A waived test cannot be selected for a test run. The waiver ID displays in the Status column. View the Summary page to see the total number of tests waived in the test run. To see which tests were waived, view the Test Run Details page.

Email Notifications

Test Manager will send email notifications (using sendmail inside VMware Workbench) to a local email address based on the configuration of preferences you set. The email notifications include a link to the Test Manager, which allows you to view the Tests Dashboard remotely.

Email notification is not enabled by default; you must specifically enable it and configure it. Email addresses that you add must be local. That is, they must be behind your firewall. For example, a VMware employee must specify email addresses with @vmware.com in them. Test Manager does not attempt to resolve email addresses outside your own company.

Open the configuration page

1   From the Window menu in Workbench, select **Preferences**.

2   In the left pane of the Preferences dialog box, expand **VMware**.

3   Expand **Test Manager**.

4   Select **Notifications**.

Configure notifications

1   Select the **Enable email notifications** check box.

2   (Optional) Select the **Enable notifications for sessions in Unit-Test mode** check box.

3   Click **Add Email**.

      The Email Address pop-up window displays.

4   Type in a local email address and click **OK**.

5   (Optional) You can remove one or all of the email addresses in the text box.

      ■   Highlight an email address and click **Remove**.
      ■   Click **Remove All** to remove all of the addresses.

6   Select which Notification Events you want to receive.

There are four notification events you can receive: **when a test session completes**, **when a test requires human intervention**, and either **when a test fails** or **when a test completes** (when a test passes or fails). The last two, **when a test fails** and **when a test completes** are mutually exclusive. You can't have both of them selected at the same time. (The check boxes for these two act like a radio button. That is, when one is selected, the other one is deselected.)

7   Click **Apply** and click **OK**.

You receive email messages from Test Manager for notification events for all test sessions that you run. You can reconfigure or disable notifications at any time.

Test Manager Command-Line Interface

The Test Manager CLI is useful for managing the scripts of Test Sessions that you've recorded with the Workbench UI. The purpose of the recordings is to capture a set of tests, the input they require, and the results of the tests (pass/fail).

A recording is the superset of tests that can be run from the CLI. You can drop any of the tests from the script, but you cannot add any new tests from the CLI because you need to capture all the user input events.

Using the CLI, you can modify the Test Session scripts, save them as new sessions, and then run them from the CLI. You can also run the same recorded tests against a new test session.

See the "Test Manager Command-Line Interface Reference" on page 39.

Running a Recorded Test Session

If you selected **Record Test Session** in the Test Progress page when you initially ran the certification session, a certification record bundle was added to the session. You can replay the entire session from the command line, and it will run unattended. The name of the CLI is `vmwb-tmcli` and the first argument must always be the workspace path. To run a recorded session from the command line, use the following command:

```
vmwb-tmcli /workspace -s /workspace/TestSessions/<session>.cert -b
            /workspace/TestSessions/<session>.d/<name of crb file>.crb
```

NOTE   The bundle is saved to the root of the cert log directory. The bundle name is <certtype>-<timestamp>.crb, where the timestamp format is yyyyMMddHHmmss.

A number of conditions are checked prior to running the certification session. If any conditions are violated, and the force (-f) option was not used, you will be prompted to confirm if you want to continue. If the force option was used, the application continues without prompting you, but you risk run failures.

For information about the Test Manager Command-Line Interface, see "Test Manager Command-Line Interface Reference" on page 39.

Running Multiple Recorded Test Sessions Simultaneously

The CLI application locks the workspace when performing any operation. This means two things:

■   You can run a separate instance of the CLI application, but you must use another workspace.

■   The CLI and the Workbench UI can share the same workspace, but they cannot run at the same time.

Other Use Cases

The following are other use cases for the Test Manager CLI:

■   "Create a New Test Session" on page 39

■   "Run a Recorded Test Session" on page 39

■   "User Prompted when a Dependency Noticed" on page 39

■   "User Not Prompted when a Failure Noticed (use of -f)" on page 39

Create a New Test Session

The following example creates a new test session:

```
# vmwb-tmcli /workspace -c <new cert name>.cert -t test -h server1.example.com -u root
Using /workspace as workspace...
Created new session '/workspace/TestSessions/<new cert name>.cert'
```

Run a Recorded Test Session

```
# vmwb-tmcli /workspace -b /workspace/TestSessions/test.d/Test-20120418160341.crb -s
             /workspace/TestSessions/<new cert name>.cert
Using /workspace as workspace...
Enter password for 'root' on 'server.example.com'

Partial run started...
'Test 1' started...
'Test 1' completed: passed
'Test 2' started...
'Test 2' completed: passed
Partial run complete
```

User Prompted when a Dependency Noticed

```
# vmwb-tmcli /workspace -b /workspace/TestSessions/test.d/Test-20120418160341.crb -s
             /workspace/TestSessions/<new cert name>.cert -x "Test 1"
Using /workspace as workspace...
Test 'Test 2' has dependency 'Test 1' excluded
Are you sure you want to continue? [y/n]
y
Partial run started...
'Test 2' started...
'Test 2' completed: passed
Partial run complete
```

User Not Prompted when a Failure Noticed (use of -f)

```
# vmwb-tmcli /workspace -b /workspace/TestSessions/test.d/Test-20120418160341.crb -s
             /workspace/TestSessions/foo.cert -x "Test 1" -f
Using /workspace as workspace...
Partial run started...
'Test 2' started...
'Test 2' completed: passed
Partial run complete
```

Test Manager Command-Line Interface Reference

The entire Test Manager CLI is listed in the tables that follow. To see the commands and options at the command line, use the --help command.

Table 4-3. Test Manager CLI Commands

Command	Options	Description
-l		List all of the certification types installed on the system, and the number of hosts required for each.
-c <cert_session_name>	-t <type> -h <host> -u <user>	Create a new session of the specified type and assign it the host(s) and user values.
-b <bundle>	-i	List information about the certification record bundle.
-s <cert_session_name>	-b <bundle> [-x <test-name>] [-f] [-p <password>]	Run the certification.

Table 4-4. Recorded Test Session Options

Options	Description
–c <session>	Name for new session (will be created in <workspace>/TestSessions project).
–t <type>	Certification type (must be one listed when using –l option).
–h <host>	Name of host for new session (must be repeated for certification types that require more than one host).
–u <user>	Name of user for new session (must be repeated for certification types that require more than one host).
–b <bundle>	Path to certification record bundle (.crb) file.
–s <session>	Path to session file (must already exist).
–x <test–name>	Name of the test to exclude (optional, can be repeated - if not specified all tests in the bundle will run).
–f	Force (optional).
–p <password>	Host password (optional, can be repeated. The order must match host order in the session file).

Prior to running the certification, the application will perform the following checks:

- Verify recordVersion in bundle against .cert XML file.

- Verify test dependencies are still satisfied if test are excluded from run.

- Verify that all tests to be run have a passed result in the bundle.

- Verify that all tests to be run have unattended–ok set to true in the bundle.

If any of these checks fail, the user will be prompted if they want to continue. The application will continue automatically if the force (–f) option is used, but the user risks run failures.

Test Manager Web Client

With the Test Manager Web Client, you can view the same certification Test Sessions that are visible in the Workbench UI. You can perform most operations from either the Workbench UI or the Web Client, such as opening a test session, checking host status, entering configuration parameters, selecting tests, and running the Test Session. There is no need for a Test Session to be opened inside the Workbench UI before viewing it in the Web Client.

When an operation is completed in one location, the other location reflects the changes. For example, you can create a new certification directly from the Web Client and the Workbench UI shows the new certification in the Project Navigator.

Before launching the Test Manager Web Client, enable the web server and start the web server in the VMware Workbench UI.

Enable and Start Web Server

To enable and start the web server perform the following steps:

1   From the **Window** menu in the Workbench UI select **Preferences,** then expand **VMware** and select **Web Server**

2   Select the check box labeled **Enable web server on start up**.

3   Click **Start Server**.

Launch Test Manager Web Client

To launch the Test Manager Web Client, just point your web browser to the Test Manager in the Workbench UI. For example, the URL might look like:

<ip–addr>:<port number>/testmanager/testmanager_webclient.html

Importing and Exporting Parameters from a File

You can import your certification parameters from a file you've previously exported from another certification and moved to your workspace. This allows you to import only those parameters that apply to your specific certification instead of the whole parameter cache.

Importing Parameters from a Parameter File

1   Open the preference page in Workbench Windows menu. (**Windows > Preferences**)

2   Expand **VMware**, and then expand **Test Manager**.

3   Select **Cert Parameters**.

4   To import the parameters from a parameter file, click **Import From Param File**.

5   In the dialog box that opens, browse to the file from which to import the parameters, and click **OK**.

    The list of parameters in the file will be merged into the cache in your workspace.

You can also export your certification parameters to a file in your workspace.

Exporting Parameters to a Parameter File

1   Open the preference page in Workbench Windows menu. (**Windows > Preferences**)

2   Expand **VMware**, and then expand **Test Manager**.

3   Select **Cert Parameters**.

4   To import the parameters from a parameter file, click **Export To Param File**.

5   In the dialog box that opens, browse to the file to which to export the parameters, and click **OK**.

    The list of parameters in the file will be added to the file.

Enabling Prefilling from the Parameter Cache

A certification session may request a set of configuration parameters that you must enter interactively. The parameters must be reentered each time the session is run. Test Manager caches these certification parameters and can be configured to prefill them the next time the session is run. Test Manager saves these parameters in a local cache in your workspace directory. You can override these prefilled values when you rerun the session. Cached parameter values for one certification may apply to sessions of another certification type. If the parameter form is similar, the parameter values will be prefilled.

Enabling Parameter Prefilling

1   Open the preference page in Workbench Windows menu. (**Windows > Preferences**)

2   Expand **VMware**, and then expand **Test Manager**.

3   Select **Cert Parameters**.

4   Select the check box labeled **Enable automatic certification parameter filling**.

    This check box is a toggle that enables or disables prefilling of certification parameters. Even if disabled, Test Manager will continue to remember new form parameters, but the parameters in the configuration parameters form will not be prefilled.

5   Then click **Apply** and click **OK** to save the preference.

Importing a Parameter Cache from Another Workspace

Cached parameters from one workspace can be merged into the parameter cache of another workspace.

Importing a Parameter Cache

1   Open the preference page in Workbench Windows menu. (**Windows > Preferences**)

2   Expand **VMware**, and then expand **Test Manager**.

3   Select **Cert Parameters**.

4   To import the cache from another workspace, click **Import From Workspace**.

5   Select the file folder where the workspace cache is located and click **OK**.

The cache is merged with the cache in your current workspace.

Time Management Help for Long Running Tests

Because some of the certifications can run for a long time, you might want to let them run unattended. To help facilitate running the tests unattended, you can configure preferences such that Test Manager sends email notifications to you for various test session events. You could leave the room, knowing that you will receive an email notification when the test session ends, or if it needs human intervention. For more information, see "Email Notifications" on page 37 and "Tests Dashboard" on page 28.

Alternately, to save time, you can split your certification tests into multiple sessions that can run in parallel. You can use this feature if two conditions are met: the test suite must allow it, and you must have multiple equivalent hosts on which to run the certification tests.

After the tests run to completion, the test sessions are merged into one new merged session from which you submit the certification. For more information on how to split your certification tests and later merge them for submission, see "Splitting Certification Tests into Multiple Sessions" on page 42 and "Merging Split Sessions into One New Merged Session" on page 43.

Splitting Certification Tests into Multiple Sessions

To speed running your certification tests, if the test suite allows merging of test sessions, you can split the tests into multiple test sessions that can be run concurrently. Because Test Manager requires all tests to be run on the same host, when you merge the sessions, the Merge Existing Certifications wizard performs equivalency tests on the hosts to ensure that all hosts or sets of hosts being used for the multiple sessions are equivalent.

Tips for Creating Target Hosts for Multiple Sessions in Installation Helper

■   Run Installation Helper as usual to create your testbeds.

■   Decide how many test sessions you want to use for this certification and create the same number and type of target hosts for each test session.

■   When selecting target hosts for your test sessions, make sure they are equivalent for each session.

For help with the Installation Helper, see the *VMware Workbench Installation Helper Guide*.

Create Multiple Test Sessions

1   In the Test Manager, create multiple test sessions of the same type.

2   Select a subset of tests to assign to each session, such that all tests are covered.

3   Run the test sessions concurrently.

For information about how to create a test session, choose tests for it, and then run it, see "Starting a New Test Session" on page 29, and "Setting Up and Running the New Test Session" on page 30.

Merging Split Sessions into One New Merged Session

After all the test sessions in your multiple-session certification have successfully completed, and you are ready to submit the certification, you must merge all the sessions by creating a new merged session. Test Manager will collect the support bundles from the targets used in each of the original sessions into the new merged session. You use the new merged session to submit your certification to VMware.

Merge Test Sessions

1   To open the merge wizard, right-click in the Test Sessions pane and select **Import > VMware > Merge Existing Certifications**.

    If none of the installed certifications support this feature, a message at the top of the wizard window states:

    ```
    There are no certification types installed that support merging.
    ```

2   In the Select Merge Leader Session page, select the test category from the **Category**, the **Session Type**, and the **Certification Session** for the merge leader session, and click **Next**.

3   In the Select Additional Sessions page, select the additional sessions to be merged from the **Certification Sessions** list, and click **Next**.

4   In the New Certification Session page, fill in the **Project** name and **Session name** for the new merged session, and click **Next**.

5   In the Summary page, review the selections, and click **Finish** if everything is correct.

6   A confirmation pop-up window asks whether you want to continue.

    If you reply **OK**, Test Manager runs the equivalency tests on each of the selected test sessions to verify that the hosts are identical in each session and then gathers all of the support bundles from each session. This can take several minutes.

    If you reply **Cancel**, then the Merge Existing Certifications wizard exits without saving the new merged session and without having gathered any results. You will have to start over again when you have time to wait for the results to be gathered.

    If you replied **OK**, and the merge completes without error, you can proceed to submit your certification using the new merged session as if you had run the certification as a single session. For information on submitting your certification bundle, see "Generating a Results Package" on page 34 and subsequent sections.

    You can rerun all the tests together as one test session by selecting the new merged session and running it. For information on rerunning test sessions, see "Running an Existing Test Session" on page 33.

Recreating a Test Session by Importing Test Results from Previous Run

If you lose a `.cert` file or it becomes corrupted, but you still have the directory where the log files are kept, you can restore the results of previous runs without having to recreate the certification session and rerun the tests. The log files are stored in the `testname.d` directory.

You can re-run all or part of the tests. For example, if the system went down while the certification was running, you can use the logs to restore the results from the tests that ran successfully, and then you can restart the certification, selecting only the tests that did not complete. This can save substantial amounts of time over rerunning the entire certification.

Import previous results

1. Create a new test session with exactly the same name, same category and same type as the one that you lost.

   When Test Manager finds a `.d` directory with the same name as the new certification, it displays the Import Previous Results dialog. For example, if your test session was named `new_cert.d`, you might see the following dialog box:
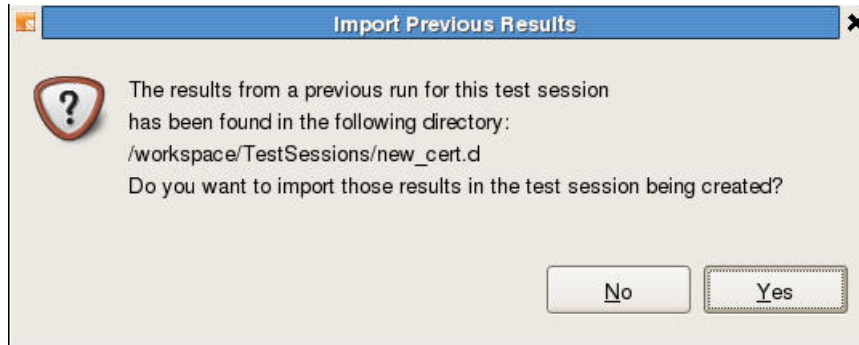
2. Click **Yes** to have Test Manager read the log files from the `.d` directory.

3. On the Host Selection page, reenter the IP addresses of the target hosts used to run the certification in the Test Manager editor, and click **Next**.

4. Click **Next** until you get to the Test Selection page.

   You must keep the same configuration and data as used in the last test run.

5. (Optional) Select the tests you want to rerun from the Test Selection page, and click **Next**.

   If you want to rerun some of the tests, you can select them while keeping the results of the other tests from the previous run.

6. (Optional) If you are rerunning some tests, click **Run Test Session** or **Run Unattended** in the Test Progress page; when the tests complete, click **Next**.

7. At the Session Summary page, you can upload the results for submission to VMware.

For information on how to generate and submit results to VMware, see "Generating a Results Package" on page 34.

Verifying Remote Installation Before You Submit Your VIB to VMware

Test your module before submitting it to VMware for certification. As a best practice:

- Install your VIB using `esxcli` software `vib install`. For more information, refer to the esxcli documentation.

- Make sure your VIB will also deploy successfully using the vCenter Update Manager (VUM). For complete instructions on using vCenter Update Manager, see the vCenter Update Manager documentation.

Before submitting your VIB to VMware for certification, as a best practice, make sure it will deploy successfully using vSphere Update Manager (VUM).

Install Your Module Using vCenter Update Manager

Use the following steps to install your module using VUM:

1. Install and configure VUM, according to VMware instructions.

2. Import the offline bundle you created for your module into the VUM package repository by selecting the "Import Patches" option and browsing to the bundle.

3   Create a baseline containing the bundle.

4   Optionally add the new baseline to a baseline group.

5   Attach the baseline or baseline group to one or more hosts.

6   Scan and remediate to install your module on the desired hosts. Update Manager will put the hosts in maintenance mode and reboot if necessary as part of the installation process.

For complete instructions on using vSphere Update Manager, see the "Installing and Administering VMware vSphere Update Manager" documentation available from VMware.

Adding A Custom Test

Though these custom tests cannot be used to certify your product, you might find it helpful to craft specific tests for your own testing purposes. Tests that you create are stored under `/opt/vmware/certs/` and are immediately accessible by the Test Manager.

To add a custom test do the following:

1   In the Workbench UI, navigate to **File > New >Other** to get the list of wizard.

2   Expand **VMware** and select **Test Template.**

Figure 4-1. Test Session Template Creation

3   In the **Test Session Template Creation** dialog, fill in the following fields:

a   **Name**: Enter the name of the session.

b   **Category**: Enter the session category, for example Networking.

c   **Number of Hosts**: The number of hosts required for the test (0-16).

NOTE   You can specify more hosts, but 16 is the recommended limit.

---

d   **Target Type**: The type of hosts: `esx` (the default), `linux windows`.

e   **Manual certification**: Check box. Select this check box if you want it to be manual certification.

f   **A text box**: Right clicking into this text box brings up the following menu:

    i   **Add Group**: Selecting this item opens the **Edit Group** dialog. Type in the name of a group and click **OK**.
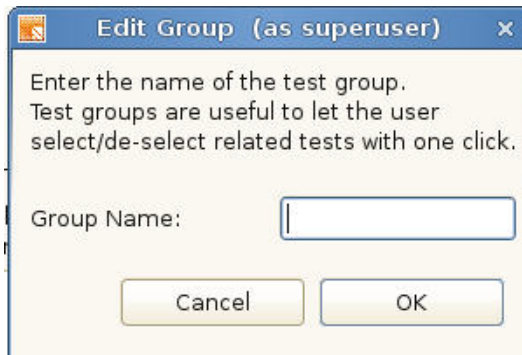
    ii   **Add Test**: Dimmed unless there is a group highlighted. Highlight a group and click Add Test to bring up the Edit Test dialog.
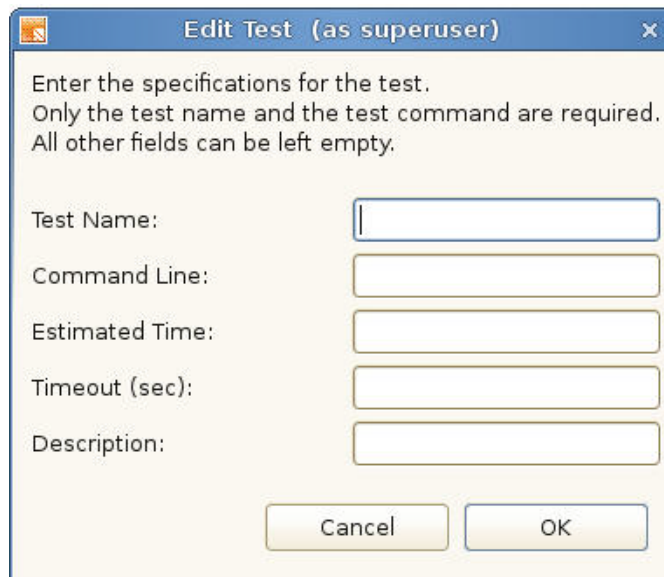
    iii   **Add Pre-Execution Test:** Add steps to be done before the test is executed. The dialog is identical to the **Add Test** dialog shown above.

    iv   **Add Post-Execution Test**: Add steps to be done after the test is executed. The dialog is identical to the **Add Test** dialog shown above.

v   **Delete Test:** Dimmed unless there is a test already added. Highlight any pre or post test, or open any group and highlight a test and click **Delete Test**.

vi   **Delete Group**: Dimmed unless there is a group to delete. To delete a group, highlight the group, right click in the box and click Delete group. You can delete the group even if there is a test inside it. No need to remove the test first. The test is lost.
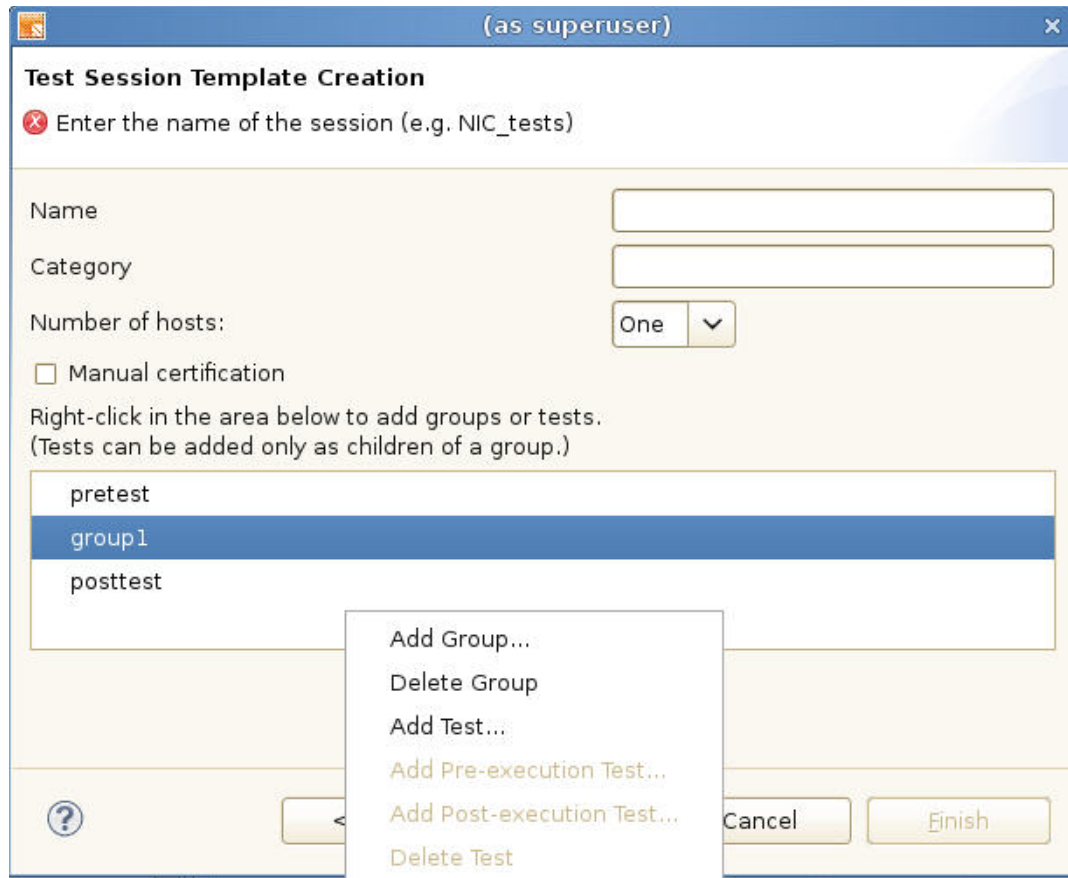
Working with Projects in VMware Workbench

5

You can create projects in VMware Workbench and then export the files to ESXi targets for testing. This chapter contains the following topics:

Creating Projects in the VMware Workbench

After you have loaded the appropriate development kit plug-ins in the VMware Workbench, start the development process by creating a project in the Project Explorer.

Open the Project Explorer

The Project Explorer opens in the C/C++ perspective. The list of open perspectives is shown in the toolbar at the top of the Workbench screen. If you are in another perspective, Workbench asks if you want to open the C, C++ perspective.

Or, you can open the C/C++ perspective by selecting **Window > Open Perspective > Other**. When the Open Perspective dialog box displays, select **C/C++** and click **OK**.

The C/C++ Eclipse perspective displays with the Project Explorer tab selected.

(Optional) Set Preference for Bringing the Build Console to View

The build console does not come to view by default when you build a project. As a one-time task, set the preference to see the build console whenever you build your project:

1 Navigate to **Window > Preferences.**

2 Expand **C/C++,** then expand **Build.**

3 Select **Console**.

4 Select the following option check box: **Bring console to top when building (if present)**.

5 Click **Apply**, then click **OK**.

Create a project

1   Right-click in the Project Explorer and select **New VMware Project/File > Development Kit Projects**.

The VMware new project wizard dialog box displays. Expand the project type you want to develop from the list of Available project types.

You can create a project from samples, which includes the infrastructure files needed for building and debugging. In some development packages, you can create a new project that does not have the sample code included (skeleton projects).

2   Expand the sample or new project type to reveal the available project types.

3   Select a project type and click **Next**.

A dialog box displays for creating the project.

4   Type a name for the project.

5   (Optional) If the **Create default xml files for VIB package** check box is active, you can choose to have a VIB created and populated with default values by selecting the check box and clicking **Next**.

Some development kits do not create VIBs at project creation, and for those development kits, the check box will be dimmed and inactive, or just not present.

If the check box is active but not selected and you do not select it, or it is selected and you deselect it, no VIB is built when the project is created. To create the VIB later, see "Using the VIB Editor" on page 51.

6   The Project Information page displays various details about the project. The page is populated with default values, but you can customize it by editing the fields.

If you selected **Create default xml files for VIB package**, click **Next** when you are finished editing the project information.

If you did not select the **Create default xml files for VIB package** check box, the **Next** button is unavailable and you are taken to the VIB Information page. This is your only opportunity to change the values of your VIB information in the Workbench UI.

7   (Optional) If you selected **Create default xml files for VIB package**, the VIB Information page displays.

The VIB fields are populated with default values, which you can customize by editing the fields.

One of the options for the VIB Information is the **Live install** flag. When **Live install** is set to **true**, firewall rules are automatically reloaded after the VIB is installed.

> ⚠ **CAUTION**   To work with the VMware Update Manager (VUM), you cannot have both **Live Install** and **MaintenanceMode** set to **false**. VUM will throw an error if both are set to false.

8   Click **Finish**.

The new project shows in the Project Explorer tab in the left pane. Expanding the project displays a list of folders created for the project. If you selected **Create default xml files for VIB package**, a VIB is created for your project. Otherwise, you can build a VIB in a separate step (see "Using the VIB Editor" on page 51).

9   To build the project, right-click on the project name, and select **Build Project**.

You must build the project before you can build a VIB.

> **NOTE**   To see the build progress, make sure the correct preferences are selected in order to bring the Build view to the front. Navigate to **Window > Preferences**, then expand the **C/C++** preference, expand the **Build** preference, and click **Console**. Make sure that both the **Open console when building** and the **Bring console to top when building (if present)** check boxes are selected. Click **OK** to save the preferences.

Using the VIB Editor

You can create a VIB for a project that does not have one, or update an existing VIB using the VIB Editor. A VIB package can be built with release objects or debug objects. If you deploy a VIB with debug objects, deploy it to a host installed with a debug build of ESXi.

For details on creating VIBs for individual development kits, refer to the appropriate Developer's Guide for that development kit in the **Help Contents**.

IMPORTANT VMKAPIDDK and DDK projects to not use the VIB Editor. Modify the *<project name>*.sc file and Makefile to configure the VIB package. For more information, refer to the *VMware VMK API Device Driver Development for ESXi* guide, or the *VMware Device Driver Development Kit Guide*.

For CIM PDK

For CIM PDK release builds

You can create a build with release objects two ways:

1   With your project highlighted in the Project Explorer, select **VIB Package (release)** from the right-click menu.

    The VIB is built with default values, but with release objects.

2   With your project highlighted in the Project Explorer, select **Create/Edit VIB xml** from the right-click menu.

    The VIB Editor view displays.

    a   Edit the fields.

        For CIM PDK, select **release** as the **VIB Build Type**.

    b   Click **Build VIB Package**.

In addition, using a file editor of your choice, you can edit the VIB XML files that were created when the project was first built.

For CIM PDK debug builds

You can create a build with debug objects two ways:

1   With your project highlighted in the Project Explorer, select **VIB Package (debug)** from the right-click menu.

    The VIB is built with default values, but with debug objects.

2   With your project highlighted in the Project Explorer, select **Create/Edit VIB xml** from the right-click menu.

    The VIB Editor view displays.

    a   In the VIB Editor, select **debug** as the **VIB Build Type**.

    b   Click **Build VIB Package**.

For information on how to debug your CIM Provider, see the *CIM Provider Development Kit (CIM PDK) Guide* found in the Workbench Help Contents.

For Host Extensions

For Host Extensions release builds

You can create a build with release objects two ways:

1 With your project highlighted in the Project Explorer, select **VIB Package** from the right-click menu.

   The VIB is built with default values, but with release objects.

2 With your project highlighted in the Project Explorer, select **Create/Edit VIB xml** from the right-click menu.

   The VIB Editor view displays.

   a Edit the fields.

      For CIM PDK, select **release** as the **VIB Build Type**.

   b Click **Build VIB Package**.

For Host Extensions debugging

There is no debug build for Host Extension plug-ins. For information on debugging a Host Extension plug-in, see "Debugging a Live Host Extensions Module" on page 76 and see the *VMware Host Extensions Developer's Guide*.

For KMDK-Based Development Kits (PSA and VDS)

For release builds

1 With your project highlighted in the Project Explorer, select **Create/Edit VIB xml** from the right-click menu.

   The VIB Editor view displays.

2 Edit the fields.

   For example, select **release** as the **VIB Build Type**.

3 Click **Build VIB Package**.

Optionally, outside the Workbench UI, you can edit the VIB XML files created when the project was first built using a file editor of your choice.

For debug builds

You can get a debug build in two ways:

1 With your project highlighted in the Project Explorer, select **Build package with debug module** from the right-click menu.

   The VIB is built with debug objects. Since you do not certify with a debug build, the VIB File Generation dialog box does not display; default values are used.

2 From the VIB Editor, select **debug** as the **VIB Build Type**.

   a With your project highlighted in the Project Explorer, select **Create/Edit VIB xml** from the right-click menu.

      The VIB Editor view displays.

   b Select **debug** as the **VIB Build Type**.

   c Click **Build VIB Package**.

Tips for VIB Fields

There are several details you should be aware of when working with the VIB Editor.

- There are multiple tabs at the bottom of the VIB Editor view:

   NOTE If you don't see one of the tabs in the VIB Editor, look in the Project Explorer under your project name and double click the file name. That will open the file in the VIB Editor.

   - **VIB Editor** – The main tab where most information can be edited.

      Clicking any of the relationship links (**Depends**, **Conflicts**, **Replaces**, **Provides**, **CompatibleWith**) or the **Software Platform** link takes you to a tab for editing the values.

   - **Relationships** – Tab that allows you to specify relationships such as `depends` and `provides`.

   - **descriptor.xml** – Tab that allows you to edit the `descriptor.xml` file directly.

   - **Bulletin Input** – Tab that allows you to edit the bulletin using a UI interface.

   - **Platforms** – Tab that allows you specify the software platforms to which this software applies.

   - **bulletin.xml** – Tab that allows you to edit the `bulletin.xml` file directly.

- The default VIB Build Type is **release**. For debugging purposes, select **debug** as the VIB Build Type.

   If you choose to create a **debug** build, deploy this VIB to a host with a debug build of ESXi 5.5 installed, not a release build of ESXi 5.5.

- The `LiveInstall` flag is set to `true` by default. To set it to `false`, deselect the **Live install allowed** check box.

   NOTE Just as with a regular install and reboot, firewall settings are automatically reloaded upon completion of a live install.

- The `LiveRemove` flag is set to `true` by default. To set it to `false`, deselect the **Live remove allowed** check box.

- You can enable maintenance mode by selecting the **Enable Maintenance mode** check box.

   CAUTION If you are using VUM to install your VIB package, you cannot set both `MaintenanceMode` and `LiveInstall` to false. VUM will throw an error.

- For CIM projects, if you are doing a live install, the **Restart CIMOM** check box should be selected (it is the default).

   The live install process does not automatically restart the CIMOM unless this is selected.

- Select the **Stateless Ready** check box if you are confident that your module functions properly in a stateless environment. That is, it does not require state information, or you have provided a host profile to collect and save the required state information. An ESXi host must be able to reboot at will and not have to save anything, and your state data cannot be recorded in a local file system.

   One relatively easy way to test that your module is stateless ready is to follow the procedures in the "Auto Deploy Proof of Concept Setup" and "Testing Stateless Readiness with Auto Deploy" sections in the *vSphere Installation and Setup* manual at `www.vmware.com/support/pubs`.

Specifying Dependencies in the VIB Editor

The ESXi VIB database specifies various functions, APIs, or features that are provided by ESXi using `provides` tags. If the item you are packaging in a VIB depends on specific features provided by the base ESXi installation, add a `depends` constraint to your VIB for each of those features.

To find out which functions, APIs, or features are provided by the ESXi base installation, run the following command on an ESXi machine:

```
esxcli software vib get –n esx–base | grep Provides:
```

If you are providing a separately bundled library or tools package that your development-kit-generated module will depend on, make sure that these separate bundles have the necessary "provides" tags. Enter "provides" constraints in the separate bundles, and use a corresponding "depends" constraint for each bundle in your development-kit-generated module.

Enter a depends constraint

1 Open the VIB Editor by highlighting your Workbench project and right-clicking **Create/Edit VIB xml**.

2 Select the **Relationships** tab and expand **relationships**.

3 Right-click **depends**, and select **Add constraint**.

4 Expand **depends**, select the **constraint**, and fill in the **constraint Properties.**

Refer to the provides tags gathered from the ESXi machine for the information to enter in the **name**, **relation** and **version** fields.

5 (Optional) Add other constraints as necessary by right-clicking on **depends**.

6 Save the changes. (**File > Save**)

Installing a VIB on a Target ESXi Machine

After creating a project, and building a VIB, you can install the VIB on a target ESXi machine. You can reach the installation wizard from either the Project Explorer, or the Remote System Explorer right-click menu.

IMPORTANT This section gives generic instructions on how the interface works. Refer to the appropriate development kit guides for specific instructions on installing your modules.

Install the VIB

1 Highlight the VIB you wish to install in the Project Explorer.

For kernel modules and host extensions modules, the VIB is located in the following directory:

`<project_name>/build/`

For CIM providers, the VIB is located in the following directory:

`<project_name>/build/obj/esx/staging/`

For device drivers, the VIB is located in the following directory:

`<project_name>/build/vib/`

2 In the right-click menu, select **Install Package**.

The Install Wizard displays.

3 Select the check box next to the target ESXi host.

You can install the package on multiple ESXi targets. If you have more than one ESXi target defined, you can select any or all of them and the install wizard will install the package on each in succession.

If you are not already connected to an ESXi target, click **Add Target** to display the New Connection dialog box. For instructions on how to use the New Connection dialog box to connect to an ESXi, see

4 Click **Next**.

The VIB is copied to the remote system under `/tmp` and automatically installed.

If the module fails to install, make sure you are installing a later version than is already installed on the ESXi machine. Check the kernel logs for detailed messages to help isolate the problem.

5 Click **Install**.

A pop-up window informs you that once installation of the VIB starts, you cannot cancel the operation. Click **OK** to continue.

Installation progress and status is written to the text box.

When you created the VIB, you had the opportunity to choose whether to do a live install. If you chose a live install, then at installation, the ESXi machine is automatically restarted and the VIB payload is loaded.

6   When the installation is complete, the **Installation Result** displays. Click **Next**.

The **Installation Summary** page appears.

7   (Optional) If you did not choose live install, then you can reboot the machine now or wait until a later time. To reboot now, select the box with the name of the ESXi host.

8   Click **Finish**.

NOTE If you explicitly rebooted or did a live install and it restarted automatically, the connection you had to the ESXi host will be broken, and you will have to reconnect using the RSE. For instructions on connecting to a remote host, see "Connecting to an ESXi Server" on page 60.

---

9   (Optional) Verify that the module loaded.

Launch a terminal by right-clicking **Ssh Terminals** under the ESXi connection in the Remote Systems Explorer. Select **Launch Terminal**.

At the command line, type the following command:

```
# esxcli system module list | grep <module-name>
```

For example for a vStorage module, you might see the following, where you get the module name, followed by true/false twice. The first for it is loaded and the second for if it is enabled: The the example that follows, the module is loaded and enabled.

```
# esxcli system module list | grep satp-example
satp-example        true      true
```

For Userworld Applications Only: VIBs Developed for Live Install/Upgrade

If you develop Userworld application VIBs for live install/upgrade and you change the host state or configuration using non-`esxcli` commands, then you will have to manually refresh the `hostd` cache for the particular subsystems involved. That is, you have to manually refresh the cache if you use `esxcfg` commands, a custom `esxcli` extension, or any command outside of the built-in `esxcli` commands. If you use `esxcli` commands, `hostd` will be able to refresh its caches automatically.

To refresh the `hostd` caches manually, use one or more of the following commands, depending on what you need to refresh:

```
vim-cmd hostsvc/refresh_firewall
vim-cmd hostsvc/refresh_services
vim-cmd hostsvc/storage/refresh
vim-cmd hostsvc/net/refresh
vim-cmd hostsvc/datastore/refresh
```

Creating Offline Bundles

When you create a project, if **Project > Build Automatically** is selected, or when you click the **Build VIB Package** button in the VIB Editor, an offline bundle is created. Bundle contents can be found in the following project directory:

```
<project name>/build/bundle/
```

For further information about offline bundles for your project type, see the documentation for your specific development kit. Also, see the *VIB Tools Guide* in the **Help Contents**.

Exporting Project Files

You can export project files to ESXi targets using the Remote File Deploy export wizard.

Export project files

1   With your project highlighted in the Project Explorer, right-click to open the Export dialog box.

2   Expand **VMware** and select **Remote File Deploy**.

3   In the left pane at the top, select the project to export.

4   Select the file or files to be exported.

    You can filter for a specific file type to select by clicking **Filter Types** and selecting the suffix. You can also select all the files displayed by clicking **Select All**, or deselect all the files displayed by clicking **Deselect All**.

5   Type in the target location for the files, for example: `<esxi-host-name>:/root/vib`, or click **Browse** to select a target machine and file path.

6   Select either **Create Directory structure for files** to create the directories from the top of the project, or **Create only selected directories** to create only the directories necessary for the selected files.

7   Type in a name for the new operation and click **Save** to create the operation, and click **Finish** to perform the operation and close the dialog box.

    The operation executes and the files are copied to the ESXi machine. The operation is saved in the Workspace with the name you gave it for future reuse.

Project Auto Deploy Settings

After creating and building a project, you can create auto deploy file operations that will be executed after every build. You can deploy your project files, and execute other commands on the ESXi machine using these operations.

Create auto deploy operations

1   In the Project Explorer, right-click on the project name and select **Properties**.

2   In the Properties dialog box, select **Auto Deploy Settings**.

3   Click **New** to bring up the Export Settings dialog.

4   Expand your project name in the left pane.

    The file names display in the right pane.

5   In the right pane, select the files to deploy.

    You can filter for a specific file type to select by clicking **Filter Types** and selecting the suffix. You can also select all the files displayed by clicking **Select All**, or deselect all the files displayed by clicking **Deselect All**.

6   Type in the target location for the files, for example: `<esxi-host-name>:/root/vib`, or click **Browse** to select a target machine and file path.

    IMPORTANT   See the VIB Suite documentation for a list of allowed directories where your files can be installed. Do not install your binaries to `/sbin`, which is a restricted directory.

7   Select either **Create Directory structure for files** to create the directories from the top of the project, or **Create only selected directories** to create only the directories necessary for the selected files.

8   Type in the name for this deploy operation.

9   Click **Save** and click **Cancel** or **Finish**.

    The operation is saved and displays in the list of available operations.

10 Select one or more operations and click **Select** to put them into the After build list.

11 (Optional) You can reorder the operations in the After build list by selecting an operation and clicking **UP** or **DOWN**.

12 Click **Apply** and click **OK** when you are finished.

These operations will then be executed after every build.

Recall and alter an existing operation

Operations you have already named and saved can be recalled and redefined.

1 In the Save/restore deploy operation section of the Export Settings dialog, click the drop-down menu.

2 Select one of the named operations in the list and click **Restore**.

3 Select the new files you want to put in this deploy operation and select the new target location, if desired.

4 Click **Save**.

A new set of files is now associated with the named operation.

Move operations from the After build list

1 In the Project Explorer, right-click on the project name and select **Properties**.

2 In the Properties dialog box, select **Auto Deploy Settings**.

3 In the After build list, select the operation to remove and click **Deselect**.

The operation is removed from the After build list, but is available to be selected at a later time.

4 Click **Apply** to save the setting and click **OK** to close the dialog box.

Remove operations permanently from the list of available operations

1 In the Project Explorer, right-click on the project name and select **Properties**.

2 In the Properties dialog box, select **Auto Deploy Settings**.

3 Select the operation you wish to remove and click **Remove**.

4 Click **Apply** to save the change and click **OK** to close the dialog box.

Add Pre- and Post-Deploy Commands to an Auto Deploy Operation

1 In the Project Explorer, right-click on the project name and select **Properties**.

2 In the Properties dialog box, select **Auto Deploy Settings**.

3 Select the operation and click **Edit**.

4 The Deploy Commands dialog displays.

Edit existing commands or add pre- and post-deploy commands to the deploy operation. Each command works as an individual shell process.

5 Click **Finish** when you are done editing the operation.

6 Click **Apply** and click **OK** to close the dialog.

Using the Remote System Explorer 6

The Remote System Explorer (RSE) is an Eclipse plug-in for connecting to and performing operations on remote systems. VMware has extended the standard RSE tool to connect to remote servers. This chapter contains the following sections:

RSE Functions

You can use the ESXi connections that you create in RSE to do the following functions:

- Transfer files between your local system and a remote system.
- Run commands on the remote system.
- Install and deploy VIB packages on the remote system
- Test your module before certification
- Run certification tests

Configuring SSH on Your Target Machine

To ensure that SSH is configured properly to work with RSE, do the following:

1   Make sure that SSH is enabled on the target machine.

    SSH is not enabled by default on a hardened SLES11 SP3 VM.

    a   In a terminal window on the target machine, log in as `root`.

    b   Start SSH.

```
service sshd start
```

2 For hardened SSH servers, change or add configure parameters.

Default values for a hardened SSH server (sshd) may be different than on other non hardened servers. Change the SSH configuration file to contain the following configuration parameters.

a Change to the location for the configuration file. The Linux configuration file is located at: `/etc/ssh/sshd_config`

b Add or change to the following values:

```
MaxSessions 20
AllowTcpForwarding yes
```

Accessing the RSE

When you open the Project Explorer by selecting **Develop Your ESXi <Module Type>**, in the development kit menu, VMware Workbench also opens the RSE.

To explicitly open the RSE in VMware Workbench, perform one of the following:

■ Select **Window > Open Perspective >Remote System Explorer**.

■ Select **Window > Show View > Remote Systems**.

Connecting to Windows Machines and Virtual Centers

In the RSE New Connection wizard, you can choose to make remote connections to Windows machines and Window Virtual Centers. The process is essentially the same as with ESXi servers as explained in sections of this document that follow, but you must use Windows login credentials.

IMPORTANT For Windows Server 2008, the Datastore daemon is not installed by default. You must install it on your Windows system and have it running before the RSE can connect to it successfully.

Connecting to an ESXi Server

You need to connect to ESXi servers for testing and debugging purposes. Use the RSE to add server connections to your VMware Workbench session. If you have not already added the ESXi machine to your list of remote hosts, add it now. If you already have the ESXi machine added to the RSE, you must connect to it before you can install modules.

Add an ESXi Server

1 Display the New Connection dialog box.

With the RSE open in the VMware Workbench, right-click inside the Remote Systems pane and select **New > Connection** from the menu. (If it is the first time you've opened the RSE, and nothing is already selected, the right-click menu will have only two selections: **New Connection** and **Import Connections**.

2 Select **ESX** from the list of system types and click **Next**.

You can filter your choices by typing **ESX** in the **System type** text box before selecting **ESX** in the list.

3 If the Parent profile drop-down menu has more than one choice, select the profile to use.

4 Supply the host name or IP address in the **Host Name** text box, or choose one of the ESXi hosts in the drop-down menu (if present).

If you are creating a new connection to a host that is not in the drop-down menu, the **Next** button becomes available. (Click **Next** to view the messages.)

5 (Optional) Type in a name in the **Connection name** text box, if you want it to be different than the host name.

6   (Optional) Type in a description of the ESXi host in the **Description** text box.

7   Select or deselect the **Verify host name** check box.

The check box is selected by default. When selected, it checks to see if the host name is a valid name.

8   (Optional) Type in a **User Name** and **Password** in the text boxes.

The `root` user name is the default. You can manage user names and passwords in the Preferences dialog box. See "Managing Passwords" on page 67 for instructions on setting and changing passwords.

9   Select or deselect the **Save password** check box.

The check box is selected by default.

10   Click **Finish**.

Start the SSH daemon on your remote system

1   In the Remote System Explorer, right-click the target ESXi machine, and select **VMware > Configure SSH Service** from the menu.

2   In the SSH configuration dialog box, observe the **Status**.

3   (Optional) If the Status is **Stopped**, click **Start** in the Service Commands area and click **OK**.

Connect to an ESXi server

Once you have added an ESXi machine, you must connect to it. With the remote ESXi host highlighted in the RSE pane, select **Connect** from right-click menu.

NOTE   If the connection fails with "failed to connect sshd to <the ESXi host>", the SSH process must be started.

Disconnect from an ESXi Server

If you wish to break the connection to one of the ESXi machines, select the ESXi host in the RSE pane and select **Disconnect** from the right-click menu.

Properties and Details For ESXi Servers

After you have connected to an ESXi server, the Remote Systems navigation tree contains many items of information about the server. The following sections describe each of the items available for ESXi servers.

Remote Systems Navigation Tree

The **Remote Systems** navigation tree contains the following items for a connection to an ESXi server.

Table 6-1. Remote Systems Navigation Tree Items

Navigation Tree Item	Description
File Explorer	Expand this item to view the local file system on the remote server.
Ssh Terminals	Click this item to create a SSH terminal window to the remote server. See "Launching SSH Terminals" on page 65 for more information.
Resource Explorer	List of resources that you can add to the ESXi machine.

Remote Systems Properties

The **Properties** tab displays information about the currently selected remote system (or **Local**).

Table 6-2. Properties for the Items in the Remote Systems Navigation Tree

Property	Description
Connection status	Connection status of subsystems.
Default User ID	User ID specified when the connection to the remote system was established.
Description	Description of this connection.
Host name	Host name or IP address of the remote system.
Name	Name of resource.
Number of children	Number of children currently under this parent.
Parent profile	Profile that owns this connection.
Remote system type	Type of remote host, for example, ESX.
Type	Type of resource, such as `Connection`.

Remote Systems Details

The Remote Systems Details tab displays the status of the connections in the Remote System Explorer. This tab provides you with an overview of the properties and other information of all connected remote systems. The columns in this tab correspond to the properties listed in Properties tab for the currently selected remote system.

Table 6-3. Remote Systems Details Tab Items

Column	Description
Resource	Name of the remote system connection.
Parent Profile	Parent profile specified when the remote system connection was established.
Remote system	Type of remote host, for example, ESX.
Connection status	Connection status of subsystems.
Host Name	Name or IP address of the remote host.
Default User ID	User ID specified when the connection to the remote system was established.
Description	Description of this connection.

Resource Explorer

The Resource Explorer displays resources and their properties for each ESXi connection in the Remote Systems tab.

Resource Types

The following resource types are listed under every remote system: Virtual Machines, SCSI LUNs, Storage Adapters, Data Stores, Virtual Switches, Network Adapters. Each resource type can be expanded to show the list of actual resources found on this ESXi machine.

When you click a resource type, a list of properties and their values show in the Properties tab. If the Properties tab is not displayed on the UI, you can access the properties by selecting the resource type and selecting **Properties** from the right-click menu.

The **Resource Explorer** itself has the following properties:

Table 6-4. Resource Explorer Properties

Property	Value
Connected	Yes/No – Answering the question, "Currently connected to this service?"
Name	Name of resources, for example, `Resource Explorer`.
Number of children	Number of entries under this resource type.
Port	Port to use when connecting to this remote subsystem.
Type	Subsystem: `Resource Explorer`
User ID	User ID for connecting to this service, for example, `root (Inherited)`
Version	Version, release and modification of remote system, if available.

Every resource type has the following properties:

Table 6-5. Resource Type Properties

Property	Value
Connection-private	Yes/No – Answers the question, "Is this a connection-private filter pool, which only exists for this connection?"
Name	Name of resource type, for example, `Virtual Machines.`
Number of children	Number of resources currently under this resource type.
Number of filter strings	Number of filter strings that are contained in this filter.
Parent filter	Filter containing this filter. Not applicable for resource types.
Parent filter pool	Filter pool that directly or indirectly contains this filter.
Type	The type of resource, for example, Remote system filter.

Resources

When you click one of the resources listed under the resource type, a list of properties and values for that resource displays in the Properties tab. The following is a list of resource types and their properties:

■ Virtual machines

The Properties tab shows the following properties for each virtual machine on the ESXi host:

Table 6-6. Virtual Machines Properties

Property	Value
File	VMX file location.
Full Name	Full resource name in system.
Guest OS	Name of guest operating system as it is identified in the system, for example, `otherLinuxGuest`.
HostName	Full host name, for example, `host1.server1.example.com`. Available only while the host is running.
IP Address	Available only while the host is running.
Memory	Amount of memory this virtual machine has, in MB.
numCPU	Number of CPUs this virtual machine has.
Power	Status of virtual machine, either `poweredOn`, `poweredOff`, or `suspended`.
Tools Status	VMware tools status.

■ SCSI LUNs

The Properties tab shows the following properties for each SCSI LUN:

Table 6-7. SCSI LUNs Properties

Property	Value
Device ID	Identifier for the device, for example, `mdx.vrrhoa2:C0:T0:L0`.
Device Type	Generic term for device type, for example, `disk`.
Model	Vendor's model number for this device, for example, `ST373207LW`.
Vendor	Name of vendor, for example, `SEAGATE`.

■ Storage adapters

The Properties tab shows the following properties for each storage adapter:

Table 6-8. Storage Adapters Properties

Property	Value
Driver	Driver name, for example `pata_amd`.
Model	Model name, for example, `AMD 8111 IDE/PATA Controller`
PCI	PCI identifier, for example, `00:07.1`
Status	Status information, if available.

■ Datastores

The Properties tab shows the following properties for each datastore:

Table 6-9. Data Stores Properties

Property	Value
Capacity	Amount of space in Gigabytes.
Free Size	Amount of free space in Gigabytes.
Location	Path for this volume, for example, `/vmfs/volumes/49b07ab9-2921257e-d158-0030485da766`.
Type	Type of file system, for example, VMFS.
VMs	List of running virtual machines that use this datastore.

■ Virtual switches

The Properties tab shows the following properties for each virtual switch:

Table 6-10. Virtual Switch Properties

Property	Value
Free Num Ports	Number of available ports on this virtual switch.
MTU	Size in bytes of the largest data unit that the switch can pass onwards.
NumPorts	Number of total ports.
Uplinks	List of vNIC (network adapter) names the virtual switch uplinks to.

In addition, each virtual switch can have multiple port groups assigned to it. Each port group has properties of its own:

Table 6-11. Port Group Properties

Property	Value
Uplinks	List of uplinks for this port group, for example, `[vmnic0]`.

Table 6-11. Port Group Properties (Continued)

Property	Value
Used Ports	Number of assigned ports in the port group.
VLAN ID	The VLAN ID for this port group. Value if no VLAN ID is assigned: 0.

■   Network adapters

The Properties tab shows the following properties for each network adapter:

Table 6-12. Network Adapter Properties

Property	Value
Driver	The name of the driver, for example, tg3.
Duplex	Duplex mode, for example, FULL, HALF.
MAC	Unique hardware identifier (MAC Address), for example, 00:30:48:5d:a7:66.
PCI	PCI identifier, for example, 02:05.0.
Speed	Link speed in Megabytes per second, for example, 100 Mbps.

To refresh a node in the Resource Explorer

1   Select a node in the Resource Explorer.

2   Select **Refresh** in the right-click menu.

To clear a bad password in the Resource Explorer

If you entered a bad password, select **Clear Password** in the right-click menu. Upon next access, the user name and password dialog box displays.

Launching SSH Terminals

With SSH enabled, you can launch an SSH terminal by selecting **Ssh Terminals** and selecting **Launch Terminal** from the right-click menu.

A Terminal tab opens with a command line available for your use for that ESXi connection.

If you added an ESXi host but you can't open a terminal, it might be a proxy issue. Check your proxy settings. See "Fixing Proxy Issues" on page 25.

Configuring SSH Services on an ESXi Host

To configure SSH services on an ESXi host, select **VMware > Configure SSH Service** from the right-click menu in the Remote Systems pane. The **SSH configuration** page displays.

Choose a startup policy

1   Select one of the startup policies: **Start automatically**, **Start and stop with host** (the default), **Start and stop manually**.

2   Click OK to exit.

Start, stop or restart SSH

1   Click one of the **Service Commands** buttons: **Start**, **Stop**, **Restart**.

The action is taken on the host immediately. You see the **Status** change.

2   Click OK to exit.

Managing Packages on an ESXi Host

You can add or remove VIBs, VIBs in an offline bundle, or image profiles to or from an ESXi host using the RSE **Package Manager**. To access the **Package Manager**, highlight the ESXi host in the Remote Explorer pane and select **VMware > Package Manager** from the right-click menu.

The **Package Manager** displays with a list of the packages installed on the host.

Add a package

1   Click **Add** to display the **Install Wizard**.

2   Select the package to install: a VIB package from a local file or URL, a VIB package from a depot or offline bundle, or an image profile from a depot or offline bundle and click **Next**.

3   Supply the location of the VIB, offline bundle or image profile.

    a   For VIBs: In the **VIB Location** page, specify the local file or URL for the VIB package to install. Click **Browse** to find it in the local file system, or click **Projects** to find it in the Project Explorer.

    b   For VIBs in an offline bundle: In the **Depot Location** page, enter a depot URL or offline bundle ZIP file from a server. Click **Browse** to find it in the local file system.

        The VIB packages available in that depot or offline bundle display.

    c   For an image profile: In the **Depot Location** page, enter a depot URL or off-line bundle ZIP file from a server. Click **Browse** to find it in the local file system.

        The image profiles available in that depot or offline bundle display.

4   Click **Next** to display the **Install Options** page.

5   Select the options for installation and click **Next**.

6   Click **Install** to start installation.

    If there are errors, they will display in the text box.

7   When the installation completes, click **Finish**.

    The Packages installed on host *<host name>* list displays with the new package listed.

Remove a package

1   Highlight one or more packages and click **Remove**.

    The **Uninstall Wizard** displays with a list of the VIB packages to be removed.

2   Select the **Uninstall Options** and click **Next**.

3   Click **Uninstall**.

    If there are any errors, they will display in the text box.

4   When the uninstallation completes, click **Finish**.

Monitoring Server Logs

Depending on which version of ESXi you have installed, a different set of logs is available for monitoring:

Table 6-13. ESX/ESXi Logs by Version

ESX/ESXi Version	Logs
ESXi 5.0, 5.1and 5.5	`/var/log/vmkernel.log`
	`/var/log/hostd.log`
	`/var/log/vpxa.log`
ESXi 4.1	`/var/log/vmkernel.log`
	`/var/log/hostd.log`
	`/var/log/vmware/vpx/vpxa.log`
	`/var/log/messages`
ESX 4.0 and 4.1 Classic	`/var/log/vmkernel`
	`/var/log/vmware/hostd.log`
	`/var/log/vmware/vpx/vpxa.log`
	`/var/log/messages`
	`/var/log/vmksummary`
	`/var/log/vmkwarning`

Monitor a server log

1   In the RSE, highlight an ESXi host.

2   Select **VMware > Monitor System Logs** from the right-click menu.

A view opens for the ESXi host logs. A default log displays.

3   To select a different log, use the drop-down menu.

The log displays. Use the scroll controls to move back and forward through the log.

Installing VIB Packages on the Remote System

Before installing a VIB on the remote ESX, make sure that `sshd` is started on the remote ESX. See "Start the SSH daemon on your remote system" on page 61.

Install package on remote system

1   To install files on a remote ESXi system, select the system host name in the Remote Systems pane.

2   Right-click in the pane, and choose **VMware > Install Package** from the right-click menu.

A dialog box displays where you specify the name of the VIB to deploy and the ESXi host to deploy it on. For instructions on installing a VIB, see "Installing Your Test VIB on a Test ESXi Host" on page 90.

Managing Passwords

To set or change user names and passwords, use the Eclipse Preferences dialog box.

To manage user names and passwords

1   Select **Preferences** from the Window menu.

2   In the left navigation frame in the Preferences window, expand the Remote Systems menu and select **Passwords**.

3   To add a user to any ESXi in the list, click **Add**.

In the New Saved Password dialog box, complete the following fields:

a   Type in the host name.

b   Select **ESX** from the System type drop-down menu.

c   Type a **User ID**.

d   Type in a password for the new user and verify it by retyping it.

e   Click **OK**.

4   To change a saved password, select a host from the host name list and click **Change**.

In the Change Saved Password dialog box, type the new password and verify it by retyping it. Click **OK**.

5   To delete a user name and password, select the entry in the list of existing host names and their user IDs, and click **Remove**.

NOTE   No confirmation is required before the entry is deleted. Use with caution.

Debugging Your Module with GDB

7

VMware Workbench includes the `gdb` debugger with which you can debug your modules. This chapter contains the following topics for debugging modules:

Live Debugging Using VMware Workbench

There are special debuggers for CIM Provider Live Debugging and CIMOM (`sfcbd`) Core Dump Debugging:

- For CIM Provider Live Debugging, follow the instructions in "Live Debugging Using VMware Workbench" on page 69 in this guide, and then see the *VMware CIM Provider Development Kit (CIMPDK) Guide* (**Help > Help Contents**) for CIM Provider-specific debugging instructions.

- For CIMOM (`sfcbd`) Core Dump Debugging, see the *VMware CIM Provider Development Kit (CIMPDK) Guide* (**Help > Help Contents**) for CIM Provider-specific debugging instructions.

For live debugging of a module that is running on an ESXi server, you must set up the debugging environment and configure both the development and test machines.

Setting Up the Debugging Environment

Your development machine (containing the VMware Workbench virtual appliance) and your testing machine (ESXi server) must be connected by a null modem serial cable, connected using COM ports.

If you are running VMware Workbench on a VMware Workstation, make sure that the serial port is accessible from the Workstation. In a terminal, type the following command and make sure the output shows the connection:

```
#ls —l /dev/ttyS0
   crw—rw—rw— 1 root uucp 4, 64 <date> /dev/ttyS0
```

Setting Up the VMware Workbench Virtual Appliance

You must also have a serial (`com`) port set up on your Workbench virtual appliance.

Make sure your Workbench virtual appliance has a serial port

1   Power off the Workbench virtual appliance.

2   In the **VM** menu for the virtual appliance, select **Settings,** and select the **Hardware** tab.

3   Check to see if there is a Serial Port device that uses COM1.

4   If there is no serial port, add one.

    a   Select **Use physical serial port**.

    b   From the drop-down menu, select COM1.

    c   Select the check box for **Connect at power on**.

5   Power on the Workbench virtual appliance.

Setting Up the ESXi Test Machine

On ESXi beta builds, both the logPort and the gdbPort are enabled by default. If you are debugging on an ESXi release build, for performance reasons, serial logging and debugging are disabled and you must enable them. In addition to enabling serial logging, you must enable breaking on the serial port (log port).

Enable serial logging and serial debugging on the ESXi machine

1   Determine the state of serial logging on your ESXi machine.

```
ESXi# esxcli system settings kernel list | grep logPort
ESXi# esxcli system settings kernel list | grep gdbPort
```

The settings are blank if serial logging is disabled, or COM1/COM2 if enabled.

2   If necessary, enable serial logging.

```
ESXi# esxcli system settings kernel set --setting=logPort --value=com1
ESXi# esxcli system settings kernel set --setting=gdbPort --value=com1
```

3   Check to see if breaking from the log port is enabled.

```
ESXi# esxcli system settings advanced get -o /Misc/BreakFromLogPortAllowed
```

4   If necessary, configure the ESXi to allow breaking from the log port.

```
ESXi# esxcli system settings advanced set -o /Misc/BreakFromLogPortAllowed -i 1
```

This change takes effect immediately without rebooting. The change is saved and persists across reboots.

5   Reboot the ESXi server for the other changes to take effect.

Testing the Serial Connection

After both the VMware Workbench virtual appliance and the ESXi test machine are configured, test the communication link between them using the following steps:

1   On the VMware Workbench virtual appliance, in a terminal, type the following command:

```
# /opt/vmware/<package name><version><build>/tools/serial rx /dev/ttyS0 115200
```

2   Reboot the ESXi machine.

```
ESXi# reboot
```

Debugging a Live Kernel Module

You start a debug session and use the **Debug Configurations** tool, as described in this section, to define how to deploy and test your project module. The setting can be saved as a named debug session and recalled and reused as necessary for debugging your application. Using the auto-deploy feature, you can separately define and save deployment operations that can be recalled and reused in other debug sessions that you create in the future.

For more information about debugging kernel modules, see *Building, Loading, and Debugging Kernel Modules* (**Help > Help Contents**, then open one of the kernel-based development kit documentation plug-ins).

Preparing for a Live Debugging Session

1   Refer to the documentation for your development kit for instructions on how to set up your debug environment.

2   (Optional) For general instructions, a cheat sheet might be available, for example, the Prepare Remote Debugging for vmkdrivers cheat sheet.

   a   **Help > Cheat Sheets**

   b   In the Cheat Sheet Selection dialog box, expand **VMware Vmkdriver Debug** and select **Prepare Remote Debugging for vmkdrivers**.

       The Prepare Remote Debugging cheat sheet displays in a pane on the right.

3   Open a connection to the target ESXi machine where your application will run in the Remote System Explorer perspective.

4   In the Project Explorer, select the project to debug.

Setting up a Live Debugging Session

1   From the Run menu, select **Debug Configurations**, or click the arrow to the right of the debug icon and select **Debug Configurations** from the drop-down menu.

2   In the Debug Configurations dialog box, double-click **VMkernel Live Debugging** to start a new debugging session with the same name as your project.

    The Create, manage, and run configurations pane displays with these tabs: Main, Debugger, Source, Common, and Auto Deploy Settings. You must fill out the fields in each tab before starting to debug.

3   Create a name for this debugger launch configuration in the Name field.

4   In the Main tab, you see the following:

Table 7-1. VMkernel Live Debugging Main Options

Option Name	Description
C/C++ Application	Name of application to debug. Default: `/opt/vmware/devkit dir/debug/objtype/vmkernel-vmvisor` where *devkit dir* is `ddk-5.5.0-<build number>` for IO devices and `kmdk-5.5.0-<build number>` for KMDK-based projects, like PSA, and *objtype* is either: `beta`, `release`, `obj`, or `opt`. Use Search Project or Browse button to select another application.
Project	Prefilled with the project highlighted in the **Project Explorer.** Use the Browse button to change projects.
Build (if required) before launching	In the **Build configuration** field, select a configuration from the drop-down menu. Then, select one of the radio-buttons: **Enable auto build**, **Disable auto build**, or **Use Workspace settings**. If you choose the latter, you can optionally configure the launch settings by clicking the **Configure Workspace Settings** link, which takes you to the **Launching** preferences.
Connect process input_output to a terminal	Selected by default.

5   In the Debugger tab, set up the debugging options.

   a   Choose the debugger type to use.

       Depending on the debugger type you choose, you see a different set of Debugger Options.

   b   To set a stop point on startup, select the **Stop on startup at** check box and specify the symbol where the program stops.

       In the text box to the right of **Stop on startup at**, the name of the default entry point (main) displays. You can choose to type a different entry point name in the text box.

c (Optional) Click **Advanced** to select or deselect two values for automatic tracking, Variables and Registers.

The check boxes are selected by default. You can deselect one or the other. Click **OK** to change settings, or click **Cancel** to leave them the same.

d In Debugger options, Main tab, set up the GDB debugger:

For the default **VMkernel GDB server** debugger, you see the following options:

Table 7-2. VMkernel GDB Server Options

Option Name	Description
GDB debugger	Default: `/opt/vmware/devkit dir/debug/vmkgdb.sh` where *devkit dir* is `ddk-5.5.0-<build number>` for IO devices and `kmdk-5.5.0-<build number>` for KMDK-based projects, like PSA. Use the Browse button to select your own script.
GDB command file	Default: `/opt/vmware/devkit dir/debug/gdb.cmd` where *devkit dir* is `ddk-5.5.0-<build number>` for IO devices and `kmdk-5.5.0-<build number>` for KMDK-based projects, like PSA. Use the Browse button to select another command file. Note: Some commands in this file can interfere with the startup operation of the debugger, for example `run`.
GDB command set	Default: **VMkernel GDB protocol**. No other choice given.
Protocol	Default: **mi** Chose alternative from drop-down menu: **mi1** or **mi2**
Verbose console mode	Check box is deselected by default.
Use full file path to set breakpoints	Check box is deselected by default.

e In Debugger Options, Shared Libraries tab, the directories listed should include all the paths that the debugger needs to find the symbol image files for the VMkernel modules and vmkdrivers. You can add more directories as needed. You can also order the directories by using the UP and DOWN buttons.

To select libraries from which to load symbols automatically:

i Highlight a directory in the **Directories** text box.

ii Click **Select From List**.

iii In the Select Libraries dialog box, select the library or libraries from which to load symbols automatically and click **OK**.

For the VMkernel GDB server debugger, you see the following options:

Table 7-3. VMkernel GDB Server Debugger Shared Libraries Options

Option Name	Description
Directories	Directories the debugger needs to scan to find symbols.
Load shared library symbols automatically	Default: Selected.
Stop on shared library events	Default: Deselected.

f In the Debugger Options tab, click the Connection subtab which is available only for the VMkernel GDB server debugger and the gdbserver Debugger. Verify that the connection type matches the one you are using to connect to your ESXi target. There are two connection types: Serial and TCP. Each type has its own options.

Table 7-4. Connection Types for VMkernel GDB Server Debugger and gdbserver Debugger

Serial Type Options	TCP Type Options
Device	**Host name or IP address**
Default: `/dev/ttyS0`	Default: `localhost`
You can enter any serial device.	You can choose any valid host name or IP address.
Speed	**Port number**
Default: `115200`	Default: `10000`
Other values: `9600`, `19200`, `38400`, `57600`	You can enter any valid port number.

6   In the Source tab, in the Source Lookup Path, select the source path.

7   In the Common tab, complete the following information:

Table 7-5. VMkernel Live Debugging Common Tab Options

Option	Description
Save As	Select **Local file** or **Shared file**. Click **Browse** to select the directory in which to save the debug session.
Display in favorites menu	Select **Debug** or **Run** to see those commands in the favorites menu.
Console Encoding	**Default: inherited (UTF-8)** For other encodings, select from the **Other** drop-down menu. Choices are the following: **ISO-8859-1**, **US-ASCII**, **UTF-16**, **UTF-16BE**, **UTF-16LE**, **UTF-8**.
Standard Input and Output	**Allocate Console (necessary for input)** is automatically selected. You can select **File** as an alternative, or in addition to the console. Click **Workspace**, **File System**, or **Variables** to create the file list. If you select **File**, you can also select **Append** to have the output appended to the file you have chosen, rather than overwriting the contents.
Launch in background	Default: selected. You can deselect this option to have the debug session run in the foreground.

8   In the Auto Deploy Settings tab, you can create file deploy operations or copy them from existing operations in your Workspace. These operations apply to the VIB you want to debug.

■   Copy from existing operations

If you already defined some file deploy operations with the Remote File Deploy export wizard, they are saved in your Workspace. To retrieve saved file deploy operations, click **Copy From** to display the Select Operation dialog box. Select the operation to copy from the list in the left pane and click **OK**.

■   Create new file deploy operations

To create new file deploy, click **New Operation** to display the New Deploy Operation dialog box.

  i   In the left pane at the top, select the files to be deployed.

      For example, select a VIB. The file names display in the right pane. You can filter for a specific file type to select by clicking **Filter Types** and selecting the suffix. You can also select all the files displayed by clicking **Select All**, or deselect all the files displayed by clicking **Deselect Al**l.

  ii   Type in the target location for the files, for example: `<esx-host-name>:/root/vib`, or click **Browse** to select a target machine and file path.

  iii   Select either **Create Directory structure for files** to create the directories from the top of the project, or **Create only selected directories** to create only the directories necessary for the selected files.

  iv   Type in a name for the New Operation and click **Finish** to create the operation and close the dialog box.

      The name of the operation displays in the list of Auto deploy operations. Repeat the copy or creation steps until you have created all the operations for this Debug Configuration.

9   (Optional) To create commands to be invoked before and after the file deploy operation, select an operation from the Auto deploy operations pane and click **Edit** to open the Edit Commands dialog box.

You can create pre- and post-deploy commands. Type them into the appropriate text box. For example, in the **Pre-deploy commands** text box, you might type in the following command:

```
#These commands are invoked before the file is copied.
#Cleanup
rm —rf /root/vib
```

In the **Post-deploy commands** text box, for example, you might type the following command:

```
#List the VIB
/etc/profile ; esxcli software vib list
```

When you have finished, click **OK** to return to the Auto deploy operations pane. You can also choose to clear all of the commands you have entered in the boxes by clicking **Clear All**.

10   Click **Apply** to save the list, or click **Revert** if you want to start over.

11   When you have finished with all of the Debug Configuration options and you are ready to start the live debugging session, click **Debug**.

The **Debug** button is dimmed if any of the necessary fields have not been filled in. Look for messages with hints about what is missing in the top banner under the page title. You will see an icon of a red circle with a white X inside preceding the hint messages, for example:

See the *Workbench User Guide* for more information on debugging in Eclipse. For more information on debugging for your development kit, see **Help > Help Contents** for a development kit-specific guide.

12   (Optional) Fix a kernel image mismatch so the debug symbols can be loaded.

Before loading the symbol information and starting the debug session, Workbench checks for a kernel mismatch between the debugger and the target ESXi host.

A common problem when debugging VMkernel code is having mismatched kernel images between the debugger and the target ESXi host. A mismatch can occur because the debugger has access to the image included in the development kit package, but the image running on the target ESXi host comes from a different build. When there is a mismatch, the debugger fails to find valid symbols.

When you click **Debug**, Workbench checks the debugger image and the target ESXi host image and notifies the user if the images do not match (see Figure 7-1).

**Figure 7-1.** Mismatched Symbols Warning
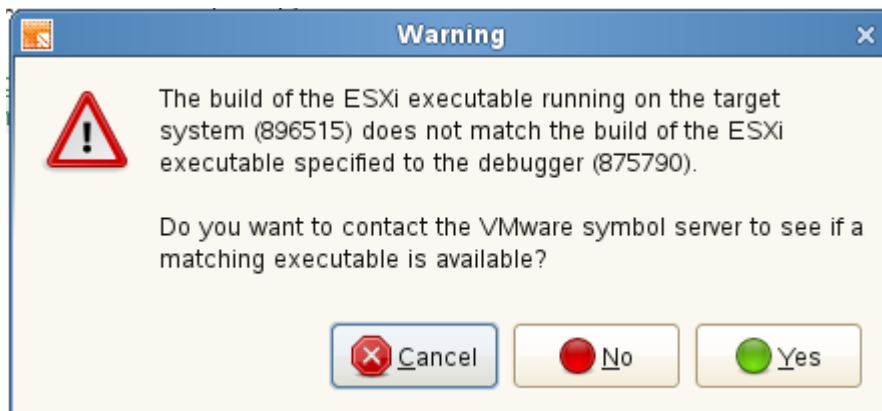
VMware maintains a symbol server from which a matching image can be downloaded if available. Click **Yes** to have Workbench search for a match. Click **No** to continue debugging with mismatched symbols.

NOTE Click **Cancel** only if you wish to cancel the entire debugging session.

Before the symbol server can be searched, you might have to authenticate. If you already authenticated in the Dashboard, then you probably will not have to do it again. If you do have to authenticate, you will see the following screen. Use your my.vmware.com credentials.
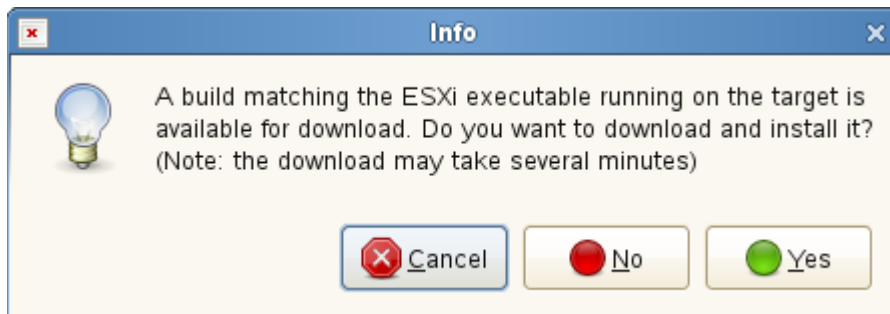
Figure 7-2. Authentication Required: Login
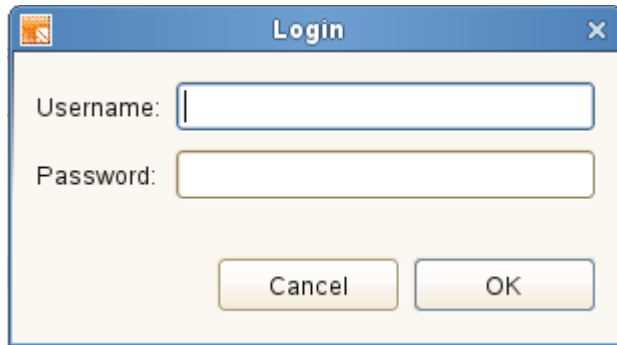
If Workbench finds a matching image, it asks for permission to install it and continue debugging.

NOTE The debug symbols file is installed in the following directory: `/opt/vmware/ddk-*`

Figure 7-3. Found a Matching Image

Click **Yes** to allow VMware to download and install the image on your debug computer. Click **No** to continue debugging with mismatched symbols. Click **Cancel** only if you wish to cancel the whole debugging session.

13 The debug perspective displays and you can proceed with debugging.

14 End your kernel debug session.

To finish the debug session and let the target ESXi host continue to run, click the disconnect button.

NOTE The terminate button is disabled for live debugging because it not only stops the test session, but also stops the ESXi and leaves it in a broken state.

Debugging a Live Host Extensions Module

You can debug your Host Extensions module using the VMware Workbench debugging tools. You start a debugging session and use the Debug Configurations tool, as described in this section, to define how to deploy and test your project module. The setting can be saved as a named debugging session and recalled and reused as necessary for debugging your application. Using the auto-deploy feature, you can separately define and save deployment operations that can be recalled and reused in your future debugging sessions.

Preparing for a debugging session

1   Refer to the documentation for your development kit for any special instructions on how to set up your debug environment.

2   (Optional) For general instructions, a cheat sheet might be available.

3   (Optional) If you are using `vmkgdb64` to connect to the live system for debugging (on a serial port), you must configure the ESXi to allow breaking from the log port.

```
ESX# esxcli system settings advanced set –o /Misc/BreakFromLogPortAllowed –i 1
```

The change takes effect immediately without rebooting. The change is saved and persists across reboots.

You can check the value to see if it is set already by issuing the following command:

```
ESX# esxcli system settings advanced get –o /Misc/BreakFromLogPortAllowed
```

4   Open a connection to the target ESXi machines where your application will run in the Remote System Explorer perspective.

5   In the Project Explorer, select the project to debug.

Setting up a debugging session

1   From the Run menu, select **Debug Configurations**, or click the arrow to the right of the debug icon and select **Debug Configurations** from the drop-down menu.

2   In the Debug Configurations dialog box, double-click **VMware Host Extensions Command Live Debugging** to start a new debugging session with the same name as your project.

The Create, manage, and run configurations pane displays with these tabs: Main, Debugger, Source, Common, and Auto Deploy Settings. You must fill out the fields in each tab before starting to debug.

3   Create a name for this debugger launch configuration in the Name field.

4   In the Main tab, you see the following:

Table 7-6. VMkernel Live Debugging Main Options

Option Name	Description
C/C++ Application	Name of application to debug (your Host Extensions command). The field is pre-filled with information from the project. Click the **Search Project** or **Browse** button to select a different application.
Project	Prefilled with the project highlighted in the Project Explorer. Use the **Browse** button to change projects.
Build (if required) before launching	In the Build configuration field, select a configuration from the drop-down menu. Then, select one of the radio-buttons: **Enable auto build**, **Disable auto build**, or **Use Workspace settings**. If you choose the latter, you can optionally configure the launch settings by clicking the **Configure Workspace Settings** link, which takes you to the Launching preferences.
Connect process input_output to a terminal	Selected by default.

5   In the Debugger tab, set up the debugging options.

    a   Choose the debugger type to use.

        Depending on the debugger type you choose, you see a different set of Debugger Options.

    b   To set a stop point on startup, select the **Stop on startup at** check box and specify the symbol where
        the program stops.

        In the text box to the right of **Stop on startup at**, the name of the default entry point (main) displays.
        You can choose to type a different entry point name in the text box.

    c   (Optional) Click **Advanced** to select or deselect two values for automatic tracking, Variables and
        Registers.

        The check boxes are selected by default. You can deselect one or the other. Click **OK** to change
        settings, or click **Cancel** to leave them the same.

    d   In Debugger options**,** Main tab, set up the GDB debugger:

        For the default VMkernel GDB server debugger, you see the following options:

Table 7-7. VMkernel GDB Server Options

Option Name	Description
GDB debugger	Default: `/opt/vmware/`*project–type*`/debug/vmkgdb64–7.2` for example *project-type* could be `PSA–<rel>–<buildnum>` for PSA devkit. Use the Browse button to select your own script.
GDB command file	Default: `/opt/vmware/`*project–type*`/debug/gdb.cmd` for example *project-type* could be `PSA–<rel>–<buildnum>` for PSA devkit. Use the Browse button to select your own script. Note: Some commands in this file can interfere with the startup operation of the debugger, for example `run`.
GDB command set	Default: **UserWorld32 GDB protocol**. No other choice given.
Protocol	Default: **mi** Chose alternative from drop-down menu: **mi1** or **mi2**
Verbose console mode	Check box is deselected by default.
Use full file path to set breakpoints	Check box is deselected by default.

    e   In Debugger Options, Shared Libraries tab, the directories listed should include all the paths that the
        debugger needs to find the symbol image files for the Host Extensions module. You can add more
        directories as needed. You can also order the directories by using the UP and DOWN buttons.

        To select libraries from which to load symbols automatically:

        i   Highlight a directory in the **Directories** text box.

        ii   Click **Select From List**.

        iii   In the Select Libraries dialog box, select the library or libraries from which to load symbols
            automatically and click **OK**.

        For the VMware Host Extensions Live Debugger, you see the following options:

Table 7-8. VMware Host Extensions Live Debugger Shared Libraries Options

Option Name	Description
Directories	Directories the debugger needs to scan to find symbols.
Load shared library symbols automatically	Default: Selected.
Stop on shared library events	Default: Deselected.

f   In the Debugger Options, Host Extensions sub-tab, you see the following options:

Table 7-9. Host Extensions Tab Options

Option Name	Description
ESXi	Specify the ESXi host name or IP address.
Port	Specify the TCP port number. The recommended range is 50000 to 50999; default is `50071`.
Location	Specify the location where the command program resides on the ESXi host. The default is `/opt/vmware/bin`.
Copy	Choose under what conditions to copy the program if it already exists at the chosen location.
Optional check box	To save the existing command program during the debugging session and restore it when the session completes, select **Backup existing program in the location on the ESXi**.
Esxcli	Specify the command line used to invoke the command program.
Optional check box	To have the debugger prompt the user to enter the command at start up, select **Ask the esxcli command line when launching the debugger**.

6   In the Source tab, in the Source Lookup Path, select the source path.

7   In the Common tab, complete the following information:

Table 7-10. VMkernel Live Debugging Common Tab Options

Option	Description
Save As	Select **Local file** or **Shared file**. Click **Browse** to select the directory in which to save the debug session.
Display in favorites menu	Select **Debug** or **Run** to see those commands in the favorites menu.
Console Encoding	**Default: inherited (UTF-8)** For other encodings, select from the **Other** drop-down menu. Choices are the following: **ISO-8859-1**, **US-ASCII**, **UTF-16**, **UTF-16BE**, **UTF-16LE**, **UTF-8**.
Standard Input and Output	**Allocate Console (necessary for input)** is automatically selected. You can select **File** as an alternative, or in addition to the console. Click **Workspace**, **File System**, or **Variables** to create the file list. If you select **File**, you can also select **Append** to have the output appended to the file you have chosen, rather than overwriting the contents.
Launch in background	Default: selected. You can deselect this option to have the debug session run in the foreground.

8   In the Auto Deploy Settings tab, you can create file deploy operations or copy them from existing operations in your Workspace. These operations apply to the VIB you want to debug.

   ■   Copy from existing operations

       If you have already defined some file deploy operations with the Remote File Deploy export wizard, they are saved in your Workspace. To retrieve saved file deploy operations, click **Copy From** to display the Select Operation dialog box. Select the operation to copy from the list in the left pane and click **OK**.

   ■   Create new file deploy operations

       To create new file deploy operations, click **New Operation** to display the New Deploy Operation dialog box.

       i   In the left pane at the top, select the files to be deployed. For example, select a VIB. The file names display in the right pane. You can filter for a specific file type to select by clicking **Filter Types** and selecting the suffix. You can also select all the files displayed by clicking **Select Al**l, or deselect all the files displayed by clicking **Deselect All**.

       ii   Type in the target location for the files, for example: `<esx-host-name>:/root/vib`, or click **Browse** to select a target machine and file path.

iii Select either **Create Directory structure for files** to create the directories from the top of the project, or **Create only selected directories** to create only the directories necessary for the selected files.

iv Type in a name for the New operation and click **Finish** to create the operation and close the dialog box.

The name of the operation displays in the list of Auto deploy operations.

Repeat the copy or creation steps until you have created all the operations for this Debug Configuration.

9 (Optional) To create commands to be invoked before and after the file deploy operation, select an operation from the Auto deploy operations pane and click **Edit** to open the Edit Commands dialog box.

You can create pre- and post-deploy commands. Type them into the appropriate text box. For example, in the **Pre-deploy commands** text box, you might type in the following command:

```
#These commands are invoked before thr file is copied.
#Cleanup
rm –rf /root/vib
```

In the **Post-deploy commands** text box, for example, you might type the following command:

```
#List the VIB
/etc/profile ; esxcli software vib list
```

When you have finished, click **OK** to return to the Auto deploy operations pane. You can also choose to clear all of the commands you have entered in the boxes by clicking **Clear All**.

10 Click **Apply** to save the configuration, click **Revert** if you want to start over, or click **Debug** if you are ready to start the live debugging session.

When the debugger starts, it prompts the user whether or not to switch to the Debug perspective.

The **Debug** button is dimmed if any of the necessary fields have not been filled in. Look for messages with hints about what is missing in the top banner under the page title. You will see an icon of a red circle with a white X inside preceding the hint messages, for example:

See the *Workbench User Guide* for more information on debugging in Eclipse. For more information on debugging for your development kit, see **Help > Help Contents** for a development kit-specific guide.

Debugging a Userworld Application in Workbench

Before you attempt to debug a Userworld application, install the gdb server. For instructions, see "Installing the C/C++ Remote Launch Feature" on page 81.

Use the Debug Configurations tool, as described in this section, to define how to deploy and test your Userworld application. The setting can be saved as a named debugging session and recalled and reused as necessary for debugging your application. Before launching a debug session, you must also configure your ESXi host.

Configure the ESX5.x remote host for gdbserver

1 Open the ports for the gdb protocol on ESXi 5.5 with the following command:

```
esxcli network firewall ruleset set –r gdbserver –e true
```

2 Disable the ASLR on the ESXi host with the following command:

```
vsish –e set /config/UserMem/intOpts/UserMemASLR 0
```

3   To debug a multi-threading application, the gdbserver requires the `libthread_db.so` library. ESXi 5.5 does not install the library. You can obtain the library from your my.vmware.com project owner. The library must be a 64-bit shared library. Install it at `/lib64/libthread_db.so.1`.

Preparing for a debugging session in Workbench

1   Refer to the documentation for your development kit for any special instructions on how to set up your debug environment.

2   In the Project Explorer, select the project that contains your Userworld application.

Setting up a debugging session in Workbench

1   From the Run menu, select **Debug Configurations**, or click the arrow to the right of the debug icon and select **Debug Configurations** from the drop-down menu.

2   In the Debug Configurations dialog box, expand **C/C++ Remote Application** to reveal your Workbench project names. Select the name of your project to start a new debugging session.

      The **Create, manage, and run configurations** pane displays with the following tabs: Main, Debugger, Source, and Common. You must fill out the fields in each tab before starting to debug.

3   Create a name for this debugger launch configuration in the Name field, or use the default, which is the project name.

4   In the Main tab, fill in the following fields as appropriate:

Table 7-11. VMkernel Live Debugging Main Options

Option Name	Description
Connection	Click **New** to add a connection to the remote server where the application to be debugged resides, or choose an existing connection from the drop-down list.
Project	Prefilled with the project highlighted in the Project Explorer. Use the **Browse** button to change project.
Build configuration	Leave this field blank.
C/C++ Application	Specify the Userworld application image file path relative to the project. The Userworld application image file is the executable and must have debug symbols or must have a link to a debug symbols file.
Remote Absolute File Path for C/C++ Application	Specify the absolute path of the Userworld application executable on the remote host.
Skip download to target path	If this check box is selected, when the debugger is launched, the Userworld application image file specified in the **C/C++ Application** filed is not copied to the remote host. If this check box is not selected, when the debugger is launched, the Userworld application image file specified in the **C/C++ Application** field is copied to the remote host using the absolute file path specified in the **Remote Absolute File Path for C/C++ Application** field.

5   Select the Debugger tab.

    a   The Debugger field is preselected to **remote gdb/mi**.

    b   (Optional) To set a stop point on startup, select the **Stop on startup at** check box and specify the symbol where the program stops.

       In the text box to the right of **Stop on startup at**, the name of the default entry point (**main**) displays. Type a different entry point name, if desired.

    c   (Optional) Click **Advanced** to select or deselect two values for automatic tracking, **Variables** and **Registers**. The check boxes are selected by default. You can deselect one or the other. Click **OK** to change settings, or click **Cancel** to leave them the same.

d   In the **Debugger Options** Main tab, set up the GDB debugger:

Table 7-12. VMkernel GDB Server Options

Option Name	Description

GDB debugger	Specify `vmkgdb64-72` with full path.
	Default path:
	`/opt/vmware/devkit dir/debug/tools/vmkgdb64-7.2`
	or
	`/opt/vmware/devkit dir/debug/vmkgdb64-72`
	depending on the development kit, and where *devkit dir* is `ddk-5.5.0-<build number>` for IO devices and `kmdk-5.5.0-<build number>` for KMDK-based projects, like PSA.

GDB command file	Change the value to be blank. Or use the **Browse** button to select your own command file.
Protocol	Use the default: **mi**
Use full file path to set breakpoints	Check box is deselected by default.

e   In the **Debugger Options** Gdbserver Settings sub-tab, specify the following options:

Table 7-13. Gdbserver Settings Tab Options

Option Name	Description
Gdbserver name	Use the default: `gdbserver`.
Port number	Specify the UDP port number for debugging. The port number must be unused and must be accepted by the firewall on the ESXi host for the gdb protocol.If the firewall on the ESXi host is configured properly for the gdb protocol, the port number range is 50000 to 50999.

6   Click **Apply** to save the configuration, click **Revert** if you want to start over, or click **Debug** if you are ready to start the live debugging session.

When the debugger starts, it prompts the user whether or not to switch to the Debug perspective.

The **Debug** button is dimmed if any of the necessary fields have not been filled in. Look for messages with hints about what is missing in the top banner under the page title. You will see an icon of a red circle with a white X inside preceding the hint messages, for example:

See the Eclipse *Workbench User Guide* for more information on debugging. For more information on debugging for your development kit, see **Help > Help Contents** for a development kit-specific guide.

Installing the C/C++ Remote Launch Feature

ESXi supports the `gdbserver` which makes it easier to debug a Userworld application. You can install the Eclipse C Development Tool (CDT) optional feature, `C/C++ Remote Launch`, on Workbench and configure it for debugging your Userworld applications.

For instructions on debugging a Userworld application, see "Debugging a Userworld Application in Workbench" on page 79. See the Eclipse help for other remote debugging information, such as remote launch configuration.

Install the C/C++ Remote Launch feature

1   Inside VMware Workbench, select **Help > Install New Software**.

2   Click the Available Software Sites link.

3   Scroll to the location of the Eclipse tools CDT URL:

    `http://download.eclipse.org/tools/cdt/releases/helios`

    The site is disabled by default.

4   Enable the site by selecting the URL (or select the check box to the left of it) and click **Enable** and click **OK**.

5   Select the URL from the **Work with** drop-down list.

    The modules available for installation are listed below.

6   Expand **CDT Optional Features**.

7   Select the **C/C++ Remote Debug Launcher** check box and click **Next**.

8   Review the **Install Details** and click **Next**.

9   Review the license notice and select **I accept the terms of the license agreement**, and click **Finish**. The installation proceeds. This step can take several minutes to complete. You cannot install the software without agreeing to the license terms.

    When the installation is complete, a dialog box advises you to restart Eclipse for the changes to take effect. Click **Restart Now** to restart Eclipse.

Debugging a Userworld Application Core Dump

The Userworld `zdump` debugger is a postmortem debugger that launches an Eclipse CDT debugger with a dump file that includes core and executable. The debugger extends the debugger services framework `gdb` launch delegate (GdbLaunchDelegate) and allows programmers specify a remote `zdump` including core file, executable image, and symbol tree directory, either interactively or automatically, to aid debugging.

Debug a Userworld program crash from ESXi host zdump

1   In VMware Workbench, make sure that the C/C++ perspective is selected.

2   Instead of Project Explorer showing your program and other items, click Remote System Explorer (RSE).

3   Right-click **New > Connection** and navigate to the ESXi host where your project's program crashed. You are asked to supply user name `root` and password. SSH must be enabled on the host.

4   Instead of RSE, click Project Explorer and return to your program. Click **Run > Debug Configurations**.

5   On the left side of the Debug Configurations window, scroll down and click **VMware User World Zdump Debugging**. New_configuration appears on the right side. You can fill out the boxes for Zdump, Core, Executable and Symbols, but this is not necessary because you can specify them later.

6   In the **Name** box for New_configuration, type a name. Click the Debugging tab. Beside **GDB debugger**, replace "gdb" by navigating to `/opt/vmware/<devkit-release#-build#>/debug/vmkgdb64*` then click OK. Beside **GDB command file**, erase the ".gdbinit" text.

7   Click the **Shared Libraries** tab. Right-click and select **Show Hidden Files** so you can see the `.build` choice in the upcoming location path. Click **Add** and navigate to this location in your workspace project: `workspace/<project>/<name>/.build/build/LIBRARIES/<devkit-project>/uw32/release` which contains debug object, stripped object, and shared object (`.so`) file. Click OK twice.

8   Click the **Zdump Main** tab and go back to the previous view. Click **Debug** on lower right.

9   The **Select Zdump File** dialog box appears. From the **Projects** pull-down menu at the bottom, select the ESXi host to which you connected RSE. On the ESXi host, select `/var/core` and click **Search**. Your `zdump` files should appear. Select one and click OK.

10 The **User World Select Symbol Tree** dialog box appears. Click **Create**. Another dialog box appears. In the pull-down menu below ESX Build Number, select the choice saying "From zdump" with a build number. Click **Next**. The Userworld symbol tree directory appears, with two check boxes underneath. Uncheck the box next to Get image files from ESX. Click **Finish** and wait for the symbol tree to be updated. When the **User World Select Symbol Tree** dialog box reappears, click OK.

11 The **Confirm Perspective Switch** dialog box appears. Click **Yes** to confirm Debug perspective.

12 In the upper left Debug pane, you can select a thread and double-click function names and line numbers to display the source code where the crash may have occurred.

13 When you return to the C/C++ perspective and the Project Explorer view, the debug project appears with your project and build files.

Debugging a Kernel Module Core Dump

The following is a general description of the core debugging facility that can be used for KMDK-based modules (PSA and DVFilter) and DDK drivers.

The VMware Workbench provides the *gdb* debugger. To open a debugging session, you must be in the C/C++ perspective with your project selected in the Project Explorer.

IMPORTANT To support the debug session, create and deploy a VIB package with debug build objects, and deploy the debug VIB package on a debug build of ESXi.

Debug a core dump produced by an ESXi system

1 From the Run menu, select **Debug Configurations**, or click the arrow to the right of the debug icon and select **Debug Configurations** from the drop-down menu.

2 Open the debugging configuration dialog in one of the two following ways:

■ In the Debug Configurations dialog box, right-click on **VMkernel Core Dump Debugging,** and select **New**.

■ Double click **VMkernel Core Dump Debugging**.

A debug configuration is created with the same name as your project.

3 In the Main tab, configure the following parameters:

a The C/C++ Application path is prefilled with a path to the VMkernel, but you can choose a different path by clicking **Search Project** or **Browse**.

b Verify the project name.

c If you want to build the project before launching the debugger, select the build configuration from the drop-down menu and select one of the radio buttons: **Enable auto build**, **Disable auto build**, or **Use workspace settings**.

If you choose **Use workspace settings**, you can configure the settings by clicking the **Configure Workspace Settings** link.

d If you do not want to connect the process input and output to a terminal, deselect the check box.

e Specify the location of the Core file.

You can click the **Search Project** or **Browse** buttons to find the core file.

4 In the Debugger tab, set up the debugger options:

a Choose the debugger type to use.

Depending on the debugger type you choose, you see a different set of Debugger Options.

b In Debugger options**,** Main tab, set up the GDB debugger:

For the default VMkernel Core Dump Debugger, you see the following options:

Table 7-14. VMkernel Core Dump Debugger Options

Option Name	Description
GDB debugger	Default: `/opt/vmware/devkit dir/debug/vmkgdb.sh` where *devkit dir* is `ddk−5.5.0−<build number>` for IO devices and `kmdk−5.5.0−<build number>` for KMDK-based projects, like PSA. Use the Browse button to select your own script.
GDB command file	Default: `/opt/vmware/devkit dir/debug/gdb.cmd` where *devkit dir* is `ddk−5.5.0−<build number>` for IO devices and `kmdk−5.5.0−<build number>` for KMDK-based projects, like PSA. Use the Browse button to select another command file.
GDB command set	Default: **VMkernel GDB protocol**. No other choices are given.
Protocol	Default: **mi** Chose alternative from drop-down menu: **mi1** or **mi2**
Verbose console mode	Check box is deselected by default.
Use full file path to set breakpoints	Check box is deselected by default.

c In Debugger Options, Shared Libraries tab, the directories listed should include all the paths that the debugger needs to find the symbol image files for the VMkernel modules and vmkdrivers. You can add more directories as needed. You can also order the directories by clicking the **UP** and **DOWN** buttons.

To select libraries from which to load symbols automatically:

i Highlight a directory in the **Directories** text box.

ii Click **Select From List**.

iii In the Select Libraries dialog box, select the library or libraries from which to load symbols automatically and click **OK**.

For the VMkernel Core Dump Debugger shared libraries, you see the following options:

Table 7-15. VMkernel Core Dump Debugger Shared Libraries Options

Option Name	Description
Directories	Directories the debugger needs to scan to find symbols.
Load shared library symbols automatically	Selected, but dimmed.
Stop on shared library events	Deselected and dimmed.

5 In the Source tab, select a source lookup path, or select other paths.

a You can choose to add other source paths and you can order them by moving them up or down in the list.

b The default path for debugging is `/project name/build`.

c (Optional) Select the **Search for duplicate files on the path** check box.

6   In the Common tab, configure the following options:

Table 7-16. VMkernel Core Dump Debugging Common Tab Options

Option	Description
Save As	Select **Local file** or **Shared file**.
	Click **Browse** to select the directory in which to save the debug session.
Display in favorites menu	Select **Debug** to see this command in the favorites menu.
Console Encoding	**Default: inherited (UTF-8)**
	For other encodings, select from the **Other** drop-down menu. Choices are the following: **ISO-8859-1**, **US-ASCII**, **UTF-16**, **UTF-16BE**, **UTF-16LE**, **UTF-8**.
Standard Input and Output	**Allocate Console (necessary for input)** is automatically selected. You can select **File** as an alternative, or in addition to the console. Click **Workspace**, **File System**, or **Variables** to create the file list.
	If you select **File**, you can also select **Append** to have the output appended to the file you have chosen, rather than overwriting the contents.
Launch in background	Default: selected. You can deselect this option to have the debug session run in the foreground.

7   Click **Apply** and click **Debug**.

Before loading the symbol information and starting the debug session, Workbench checks for a kernel mismatch between the debugger and the ESXi host that generated the dump.

You can delete all of the changes you made to the operation by clicking **Revert** instead of **Apply**.

See the Eclipse *Workbench User Guide* for more information on debugging in Eclipse (**Help > Help Contents**).

8   (Optional) Fix a kernel image mismatch so the debug symbols can be loaded.

A common problem when debugging VMkernel code is having mismatched kernel images between the debugger and the ESXi host that produced the dump. A mismatch between the two can occur because the debugger only has access to the image included in the development kit package, but the ESXi host that produced the dump might be running a different release. When there is such mismatch, the debugger fails to find valid symbols which makes debugging difficult.

When you click **Debug**, Workbench checks the debugger image against the image of the ESXi host that created the dump and notifies the user if the images do not match (see Figure 7-4).

**Figure 7-4.** Mismatched Symbols Warning
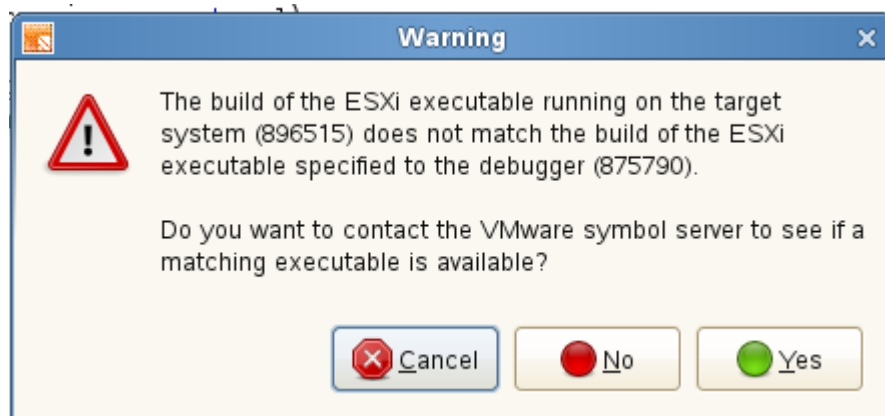
VMware maintains a symbol server from which a matching image can be downloaded if available. Click **Yes** to have Workbench search for a match. Click **No** to continue debugging with mismatched symbols.

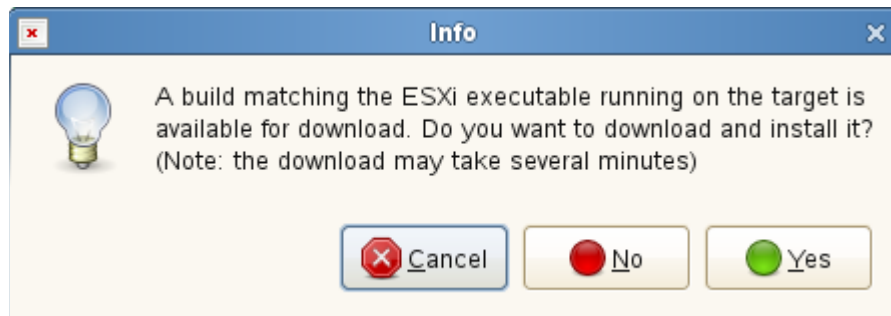NOTE   Click **Cancel** only if you wish to cancel the entire debugging session.

Before the symbol server can be searched, you might have to authenticate. If you already authenticated in the Dashboard, then you probably will not have to do it again. If you do have to authenticate, you will see the following screen. Use your my.vmware.com credentials.

Figure 7-5. Authentication Required
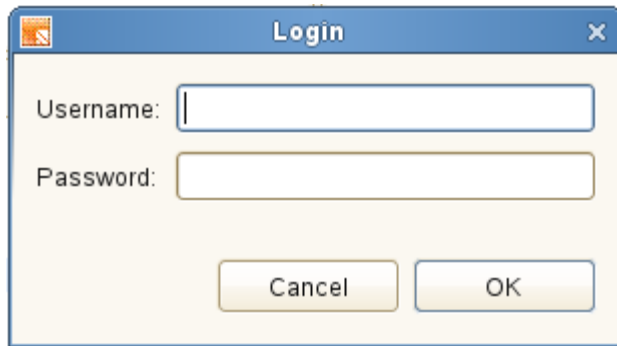
If Workbench finds a matching image, it asks for permission to install it and continue debugging.

Figure 7-6. Found a Matching Image

Click **Yes** to allow VMware to download and install the image on your debug computer. Click **No** to continue debugging with mismatched symbols. Click **Cancel** only if you wish to cancel the whole debugging session.

NOTE   The debug symbols file is installed in the following directory: `/opt/vmware/ddk-*`

---

9   The debug perspective displays after `gdb` loads the VMkernel modules and the symbolic information.

In the Debug pane at top left is a stack trace of the location of the failure. You can select different stack levels. The source file for the active stack shows in a pane under the stack trace. At the top right, the Variables view shows the local variables for the active function. Other views are available, such as views to see the current value of the CPU registers, to inspect the content of the target memory, and to see the list of breakpoints.

IMPORTANT   Depending on the currently selected stack level, the debugger may or may nor have access to the source code. You can still debug functions for which there is no source code available at the assembly level. Variables and registers should display correct values even when no source code is available.

Signing VIBs for Test and Production

<div style="text-align: right; font-size: 3em;">8</div>

To deploy your module, you must package it in a VIB. The VIB can be signed with test keys and certificates until you have certified and are given your production key and certificate from VMware.

> **NOTE** New for vSphere 6.0, during development, if you are installing an unsigned VIB to your test host using the command line, you must use the `—no—sig—check` option with the ESXi host set at the `CommunitySupported` acceptance level. When you are ready to run certification tests, you must sign your VIB, using the test keys and certificates provided by VMware, at the acceptance level for which you wish to certify.

The chapter contains the following topics:

Test Keys and Certificates

VIBtools contains the VMware test keys and certificates that you use to sign your VIBs for testing and certification and is integrated into Workbench. The test keys and certificates are located in the `/opt/vmware/vibtools/testcerts` directory. There is a set of keys and certificates for each acceptance level for which you can certify, that is, VMware Certified, VMware Accepted, Partner Certified, and Community.

> **NOTE** To obtain a VMware signed key and certificate for your production VIB, you must certify at the VMware Certified or VMware Accepted level.

With the test certificates installed on your target ESXi host, a VIB created with the appropriate test keys will be installable on the target ESXi host.

Installing Test Certificates

Your module to be tested has to be signed with the test keys as described earlier. The matching test certificates must be installed on your test ESXi host before you install your test VIB.

There are two ways to install test certificates on your target host. You can perform a fresh install on a host machine, or you can install the certificates only on an existing ESXi host. The latter is only appropriate for engineers testing their development modules and who want to install their unsigned VIBs on the ESXi machine.

- For certification and for testing signed VIBs, perform a fresh install of the special ESXi ISO, which has test certificates already installed on it:

  a   Download the `VMware–VMvisor–Installer–With–Test–Certs–6.0.0–<build number>.x86_64.iso` from the Developer Center.

  b   Install it on the test host.

- For development testing purposes only, to facilitate installing your unsigned test VIBs, you can install the VMware-provided, unsigned VIB that contains only the test certificates on an existing ESXi host:

> ⚠ **CAUTION**   You cannot raise the acceptance level on the ESXi host once an unsigned VIB has been installed on the host.

  a   Set the acceptance level to `CommunitySupported` on the ESXi 6.0 host.

  You must set the acceptance level to `CommunitySupported` level because the test-certificates-only VIB is unsigned.

  `esxcli software acceptance set ––level=CommunitySupported`

  b   Copy the `vib–test–certs` package, found on the VMware Developer Center, to the ESXi 6.0 host in the `/tmp` directory and then use the following command to install the VIB:

  `esxcli software vib install –v /tmp/vib–test–certs.vib ––no–sig–check`

  The `––no–sig–check` option is required with the `esxcli` command when installing unsigned VIBs.

Signing Your Test VIB

You can sign your test VIB using the Workbench UI or the command line.

> **IMPORTANT**   Once your module is ready to start certification testing, you must sign the VIB with the proper test key. Select the test key that corresponds to the acceptance level for which you are certifying.

The VIB can be signed using Workbench or by running the Makefile in the command line:

- "Signing a VIB Using the Workbench UI" on page 88
- "Signing a VIB in the Command Line" on page 89

Signing a VIB Using the Workbench UI

Use the following process when you are ready to sign the VIB for testing and certification purposes:

1   Create the VIB for your development kit.

See your development kit guide for instructions on how to create a VIB. (Generally, you select a VIB-creation related menu item from the right-click menu in Project Explorer.)

> ⚠ **CAUTION**   If the acceptance level in the VIB does not match the keys you are trying to use to sign the VIB, you will see the following error: `The VIB of "<level name>" acceptance level can't be signed with "<cert level name>" cert.` When you create the VIB, the default acceptance level is `community`. Be sure to set the desired acceptance level in the `descriptor.xml` file before you create the VIB. See "Selecting an Acceptance Level in the descriptor.xml File" on page 90.

2   Double-click the VIB in the Project Explorer.

This opens the VIB in a Workbench pane.

3   Select the **Signature** tab at the bottom of the pane.

4   Browse to the `testkey` directory in the local file system. (Click **Browse**)

The Select File window displays.

a   Double-click **File System** in the left pane.

b   In the right pane, browse to `/opt/vmware/vibtools/testcerts`.

5   Select the appropriate key file for your acceptance level. The available test keys are:

Table 8-1. Available Test Keys

Acceptance Level	Key Name
VMware Certified	`vmware.key`
VMware Accepted	`accepted.key`
Partner Certified	`partner.key`
Community	none required

6   Click **Browse** to select the appropriate certificate file for your acceptance level. Available test certificates:

Table 8-2. Available Test Keys

Acceptance Level	Certificate Name
VMware Certified	`vmware.cert`
VMware Accepted	`accepted.cert`
Partner Certified	`partner.cert`
Community	none required

7   Click **Add Signature**.

If you are successful, you see the following message:

`Successfully signed /workspace/<project name>/build/<vib name>.vib.`

If you see the following error message, you must edit the VIB's `descriptor.xml` file to change the VIB acceptance level to match the test key and certificate acceptance level you chose. To fix this, you can edit the `descriptor.xml` file directly in the Workbench UI.

`"The VIB of "<level name>" acceptance level can't be signed with "<cert level name>" cert."`

Signing a VIB in the Command Line

If you wish to sign your VIB with a test key in the command line instead of using the Workbench UI, install the test keys and certificates on your development machine and the target ESXi host using the instructions in the procedures that follow. After installing the test keys and certificates, a licensed VIB created with these test keys will be installable on the ESXi host.

NOTE   If the test keys and certificates are not installed on your development machine, when you try to run the license example Makefile, you will see the following error: `"The VIB acceptance level (partner) doesn't match the signing certificate."`

Install test keys and certificates on your development machine

1   Back up the following files:

```
/opt/vmware/vibtools/certs/vmware.cert
/opt/vmware/vibtools/certs/vmpartner.cert
/opt/vmware/vibtools/certs/vmpartner.crl
```

2   Replace the above files with those found in the `test-keys/devkit-test-certs` directory.

CAUTION   You must restore the original files to `/opt/vmware/vibtools/certs` before attempting to create production VIBs, or else your VIBs will be unrecognized by a production, non-test version of ESXi.

Signing Your VIB

Run the Makefile in the CLI to create a VIB with the test keys

Selecting an Acceptance Level in the descriptor.xml File

When you are ready to sign your VIB, if necessary replace the default acceptance level in the `descriptor.xml` file with one of the following levels:

Table 8-3. Descriptor.xml Acceptance Levels

Acceptance Level Name	Term to Use in the Descriptor.xml File
VMwareCertified	certified
VMwareAccepted	accepted
PartnerSupported	partner
CommunitySupported	community

The acceptance level you choose when signing your VIB (using the keys and certificates) must agree with the acceptance level in the `descriptor.xml` file. Verify that the `descriptor.xml` lists the same acceptance level you are going to use to sign the VIB. If it does not match, edit the file to list the same acceptance level.

Changing the acceptance level in the descriptor.xml file

1 With the development kit project highlighted in the Project Explorer, double-click the VIB creation choice (the choice differs by development kit) from the right-click menu.

   The VIB Editor displays in a pane in the Workbench UI.

2 Select the **descriptor.xml** tab at the bottom of the pane.

3 Find the acceptance level tag in the file.

   For example, the default setting might be as follows:

   `<acceptance–level>community</acceptance–level>`

4 Edit the tag directly in the interface and then save the file.

   Select one of the following values: `certified`, `accepted`, or `partner`.

   (You do not sign VIBs that carry the `community` acceptance level.)

   Build the VIB by clicking **Build VIB Package** in the VIB Editor.

Set the Acceptance Level on Your Test ESXi

The acceptance level on your test ESXi host must match the test key on your test VIB, and the matching test certificate must also be installed on your host ESXi host.

If you have not already done so, make sure the acceptance level on your test ESXi is the same as the test key with which your test VIB is signed. For example, you can get test host the acceptance level and the reset it:

```
esxcli software acceptance get
<prints the current acceptance level>
esxcli software acceptance set ––level=<level matching your test VIB>
```

Installing Your Test VIB on a Test ESXi Host

You can install your test VIB on a test host using Workbench or the command line.

- "Installing a VIB Using the Workbench UI" on page 91

- "Installing a Signed VIB Using the Command Line" on page 91

- "Installing an Unsigned VIB Using the Command Line" on page 91

Installing a VIB Using the Workbench UI

1. With the project expanded in the Project Explorer, expand **build** and select the VIB.

2. Select **Install Package** from the right-click menu.

   The Install Wizard displays with the file to install already selected. You can browse to a different file.

3. Click **Add Target** to choose the ESXi hosts where you will install the VIB.

   The New Connection wizard displays.

4. Select **ESXi** and click **Next**.

5. Fill in the **Host name** (the **Connection name** is filled in and mirrors the **Host name**) and optionally add a **Description**.

6. Enter the user name and password for the ESXi system and click Next or Finish to return to the Install Wizard.

7. Select the ESXi host by clicking the check box next to the host name.

   You can add multiple targets by clicking Add Target again and repeating the add target process until you have all the targets you need.

8. When you have added all the desired targets, click **Next**.

9. Click **Install**.

   Installation of the VIB package starts on the ESXi hosts you have selected. This can take a few minutes.

   When installation is complete you receive a results message in the text box. If there is a failure, an error message displays in the text box.

10. Click **Next** to see the Installation Summary for each ESXi host.

11. Click **Finish** to dismiss the installation wizard.

Installing a Signed VIB Using the Command Line

Once the VIB has been signed with the appropriate test key, install it using the following `esxcli` command:

```
esxcli software vib install
```

Installing an Unsigned VIB Using the Command Line

For development purposes only, if you wish to install your unsigned VIB to the test host, you can use the command line, as shown in the example that follows. Note that you must first set the acceptance level of the test host machine to `CommunitySupported`. When you are ready to have your module go through certification testing, you must use Workbench to sign your module with the acceptance level for which you wish to certify.

Example 8-1. Installing a Test VIB Using the -no-sig-check Option

First set the test host machine's acceptance level to `CommunitySupported`:

```
<!-- optionally, you can check to see what the level is set to first -->
esxcli software acceptance get
<!-- prints the current acceptance level -->
esxcli software acceptance set --level=CommunitySupported
```

Then install the test VIB using the `--no-sig-check` option:

```
esxcli software vib install --no-sig-check
```

CAUTION If you are installing an unsigned VIB for testing purposes, you must change the acceptance level on the test ESXi to `CommunitySupported`, or the install will fail.

Changing the host's acceptance level to CommunitySupported should only be done in a testing environment that is isolated from other hosts. Doing so in a production environment will affect the supportability of the host by VMware and partner support organizations. If you are using the CLI to install your unsigned VIB, the `--no-sig-check` option bypasses digital certificate validation of the installed package, which is allowable for your own testing environment, but in production could completely compromise the integrity of a host. This must never be done in production and must never be suggested to your customers

Signing Your VIB for Production and Distribution

After you have certified your host extension with VMware, you receive a formal key and certificate with which you will sign your VIB for production and distribution of your host extension. Use the following process for signing your VIB.

1   Store the key and certificate files in a secure location on your development machine.

2   Verify that the `descriptor.xml` file has the correct acceptance level (the one you certified for).

3   Locate and double-click the **build-bundle.inc** file for your project in the Project Explorer.

4   Edit the following lines in the `build-bundle.inc` file to include the location of your formal key and certificate files. For example, substitute the path to your key and certificate in the appropriate certification level. If your secure location path is `/your/secure/location` and your acceptance level is `certified`, the file would have the following lines:

    CERTIFIED_DEV_SIGNING_PREFIX = /your/secure/location/vmware

5   Save the file and then generate your VIB.

    For example, after saving the file, you can open the VIB Editor (right-click on the project and select **Create/Edit VIB xml**), and click **Build VIB Package**.

Verifying Remote Installation

Test that your module deploys successfully before submitting it to VMware for certification. As a best practice:

■   Install your VIB using the command `esxcli software vib install`. For more information, refer to the `esxcli` documentation found at vmware.com.

■   Make sure your VIB will also deploy successfully using the vCenter Update Manager (VUM). For complete instructions on using Update Manager, see the vCenter Update Manager documentation.

Before you distribute your Host Extension module to your customers, a best practice is to make sure it will deploy successfully using the vSphere Update Manager.

Install Your Module Using vCenter Update Manager

Use the following steps to install your module using VUM:

1   Install and configure VUM, according to VMware instructions.

2   Import the offline bundle you created for your module into the VUM package repository by selecting the "Import Patches" option and browsing to the bundle.

3   Create a baseline containing the bundle. Be sure to choose a "Host Extension" type for the baseline.

4   Optionally add the new baseline to a baseline group.

5   Attach the baseline or baseline group to one or more hosts.

6   Scan and remediate to install your module on the desired hosts. Update Manager will put the hosts in maintenance mode and reboot if necessary as part of the installation process.

For complete instructions on using vSphere Update Manager, see the "Installing and Administering VMware vSphere Update Manager" documentation available from VMware.

Using the Workbench Log Browser

9

The Log Browser in the Workbench UI supports the viewing and analysis of vSphere and Workbench log files. This chapter presents the following topics:

- "Loading Log Files into Workbench" on page 93
- "Tour of the Workbench Log Browser" on page 94
- "Recording a Log Browser Session" on page 95
- "Displaying and Using the Activity Graph" on page 95
- "More About the Log Browser UI" on page 97

Log Browser features are also available in the vSphere Web Client, however they operate faster in Eclipse or VMware Workbench, and you can choose to display an activity graph.

Loading Log Files into Workbench

The Workbench Log Browser works by creating `.logx` files in a designated folder. Rather than open log files directly, Log Browser is driven by these `.logx` configuration files, which tell Eclipse where to find a set of logs on local disk, or how to import a set of log files across the network. The `.logx` files also describe how to parse the log files into columns for display in a table.

Prerequisite To fetch logs from a remote system, if you have a network proxy set, you first must add the system's hostname or IP address to the **Proxy bypass** list in **Window > Preferences > Network Connections**.

When you open (or double click) a `.logx` file in the Project Explorer pane, a tabular display with controls appears in the same pane where the Dashboard appeared. You can then filter, highlight, and examine the log.

To start using the Log Browser

1 In VMware Workbench, click **File > New > Log Analysis**.

   Or to analyze logs associated with a project, select the project and right click **New > Log Analysis**.

2 To the right of the **Source host** drop-down list, click **Add**.

3 In the Select Remote System Type dialog, click **ESX** for an ESXi host, **Virtual Center** for vCenter Server, or another choice. You will probably be asked for a password.

4 Back in the Log Host and Format Selection dialog, choose a **Log Format**. You can either Browse to an existing **Project container**, or type in the name of a new folder to be created.

   For vCenter Server Appliance, you will need to know the parent directory where your log files reside. For example, the **vpxd** logs appear in the /var/log/vmware/vpx directory.

   For an ESXi host, you can select Standard Log Collection and get most of the available log types.

5 In the Project container area, double click (or open) one of the `.logx` files. A new middle pane appears.

To analyze Log Browser entries

1   With a log that scrolls past the end of pane, type a search term in the text box on upper left and click **Filter**. To make the filter more precise you can enable **Case sensitive**.

2   To restore all lines in the log, click **Unfilter**.

3   You can display context around a matching filter term, and show one (1) or more adjacent lines in the log, by changing **0** in the drop-down list between **Filter** and **Case sensitive**. The lines of context appear with alternating green and white backgrounds.

4   Instead of or in addition to filtering, you can emphasize a term by typing in the text box on upper right and clicking **Highlight**. The term appears in yellow.

5   There are two tabs at the bottom of the pane, one saying Log Viewer, the other giving the `.logx` file name. If you click the second, it shows the XML definition for retrieving and formatting this log type.

Tour of the Workbench Log Browser

Once you have a set of `.logx` files in a project, and open one of them, you can browse logs using directives in the **vLogBrowser** menu or Eclipse toolbar.

Opening an Existing Log Project

You can open an existing log analysis file that you saved to the Project Explorer by expanding the project name and double-clicking the `.logx` file. The file opens in a Workbench Log Browser view.

Enhanced Search with Named Filters

In addition to simple keyword filtering, you can perform more complex filtering using the Named Filter facility. With a Log Viewer pane displayed, find the Eclipse menu and click **vLogBrowser > Filter**. The toolbar contains the same icon as a shortcut.

A dialog box appears, Keyword Filtering with a Named Filter. By default the Filter Source is the current `.logx` file, but you can also open other `.logx` files, or take filters from a file on disk.

You can create a new named filter by adding keyword or regular expressions and using And/Or/Not boolean operators, then clicking **Save**. Later you can use the **Named Filters** drop-down list to choose that saved filter by name. You have a drop-down box to include 1 or more adjacent lines. When ready, click **Save and Filter**.

Only filters from this `.logx` file can be applied to the log. Filters from other sources can be viewed, but must be saved to this `.logx` file before you can edit or apply them.

Copying Log Entries to Clipboard

To search for log entries that match a search string, find the Eclipse menu and click **vLogBrowser > Find**. The toolbar contains the same icon as a shortcut.

In the Find by keyword dialog box, you can click either **Find All** or **Find next**. In either case, click **Close** then **vLogBrowser > Copy** to copy all these log entries to your desktop Clipboard.

Finding and Synchronizing Time Stamps

To see a log entry created near a certain time, in the Eclipse menu click **vLogBrowser > Find or Adjust Date.** The toolbar contains the same icon as a shortcut. In the Find by Timestamp or Adjust All Timestamps dialog, use the date and time picker to select a month and date, optionally change time, and click **Find**.

If you want to synchronize time stamps between logs, click the **Sync** button in both `.logx` browser windows, and the matching log entries that are nearest your specified time will be highlighted in both windows.

The **Sync** button also works without the dialog box. If you highlight a log entry in one `.logx` browser window a nearby timestamp will be highlighted in other `.logx` browser windows with **Sync** button checked.

You can also adjust time stamps, for instance to correct clock skew. This adjustment does not affect the actual log entires, only the display in the `.logx` browser window. To adjust time stamps, specify a date and time and click **Find**. In the box underneath Time, specify the date and time to adjust and click Adjust. The adjusted appearance appears in red, and the time difference (as an offset minus or plus) appears beneath. To undo the time adjustment, click **Clear Adjustment**.

Refreshing a Log

To reload the log entries in a `.logx` file, click **vLogBrowser > Refresh**. The toolbar contains the same icon as a shortcut. The lower right corner of Eclipse shows a progress bar for the refresh.

To view information about a loaded or refreshed `.logx` file, click **vLogBrowser > View Log Info**. The toolbar contains the same icon as a shortcut. The Log File Information dialog shows the source host, the full path and filename, the time stamp, and the file size in bytes. Log files are listed in chronological order.

If there is a log file on the local system that you want to add to the list, click **vLogBrowser > Add Log Files** and browse to the location of the local log file.

Click **vLogBrowser > Export** to export the current `.logx` file, in a variety of formats (XML, text, JSON, CSV) to a local file, in your Eclipse workspace by default. You can export all events, or a subset of filtered events.

Cheat Sheets

Eclipse offers Cheat Sheets to introduce certain features.

NOTE You can view cheat sheets for Workbench Log Browser by clicking **Help > Cheat Sheets** then expanding **VMware vLogBrowser** to select the desired cheat sheet.

VMware implemented three cheat sheets to explain Log Browser. Two of those cheat sheet topics are not covered in this chapter: importing a remote log file into the local project, and opening a local log file.

Recording a Log Browser Session

You can create a recording of some or all the actions that you take in the log browser. The recording is stored as a shell script (`lb-script.sh`) that can be replayed from the command line, edited, or embedded in another shell script. For more information about the command-line, see Appendix A, "Log Browser Command Line," on page 99.

Start and stop recording

1   Start the recording of log browser actions by clicking the red start recording icon.

2   In the **Select a file to save generated script contents** pop-up window, either accept the default name (lb.-script.sh) and location (inside your project folder), or choose your own name and location, and click OK.

   If you are attempting to overwrite an existing script with the same name and location, a warning pops up. Click **OK** to continue overwriting an old script, or click **Cancel** to rename or relocate the script.

   The icon changes to the stop recording icon.

3   After you perform the log browser actions you want to record, click the stop recording icon:

Displaying and Using the Activity Graph

The log browser includes an activity graph view that can show activity from multiple log files over a period of time. Activity graphs can be useful to identify irregular activity, such as a system reboot or a peripheral that stops responding.

Open the Activity Graph

1 From the Window menu, click **Show View > Other**.

2 From the **Show View** dialog box, expand **VMware**.

3 Click **Activity Graph** and click **OK**.

Figure 1. Activity Graph displaying three logs
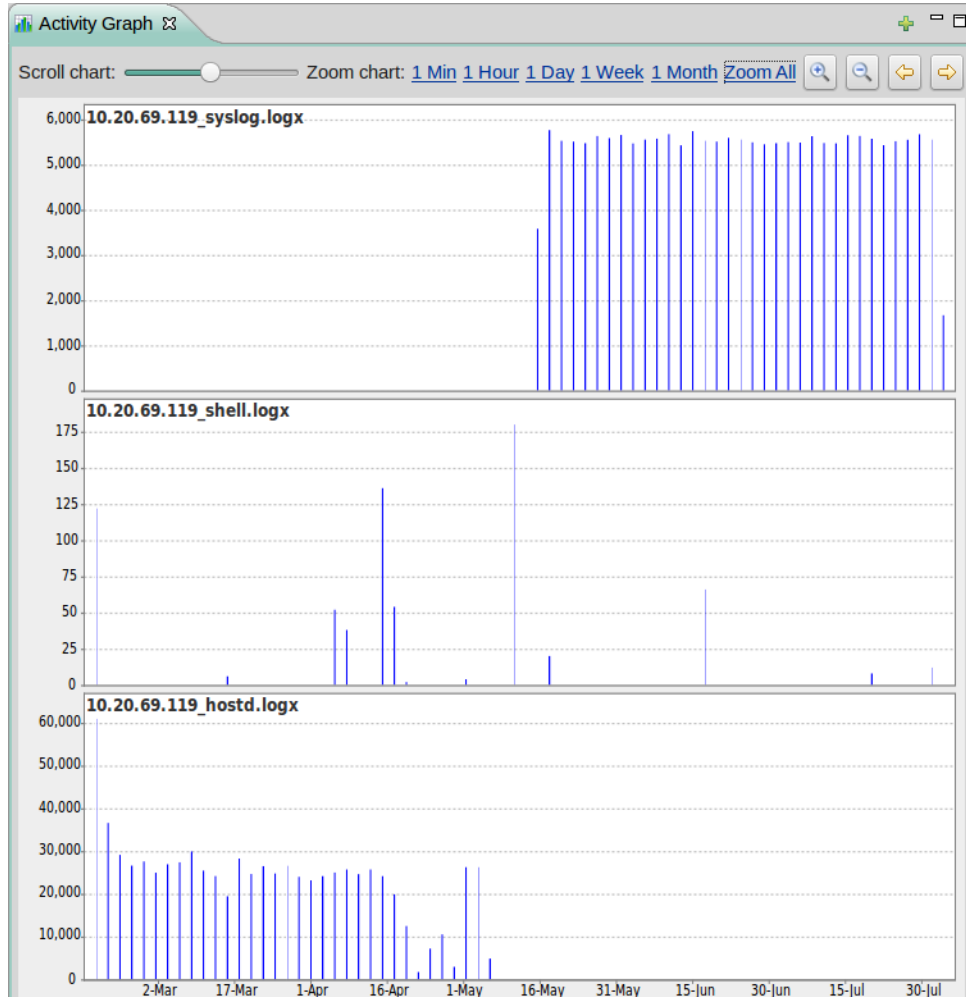
Display log files in the activity graph

1 Click the plus ("+") icon at top right of the activity graph to open the **Add Log File(s) to Graph** dialog box.

2 Select one or more log files that you want to display. Click **OK**.

The dialog box displays the log files already open in Workbench Log Browser. Each log is represented by a separate time line with the time scale spread out on the horizontal axis.

You can enlarge the activity graph window by clicking the Maximize button on upper right, and return to original size by clicking the Restore button (same location, different name and shape).

3 Adjust the time scale on the horizontal axis in one of three ways:

a In one of the time lines, click and drag the cursor across an area of the time line that you are interested in expanding. When you release the cursor, the area you highlighted is expanded to fill the entire time line. The time scale is adjusted on the bottom axis.

b Select one of the **Zoom chart** links at the top of the activity graph: **1min**, **1hour**, **1day**, **1week**, **1month**.

The time line at the bottom changes to display the activity during that time interval.

c   Click the **Zoom In** or **Zoom Out** (magnifying glass) buttons to uniformly expand the time scale or shrink the time scale.

There are upper and lower bounds to the zoom scale. You will receive a warning message if you Zoom In to less than 100 ms, or Zoom Out to more than 10 years.

4   Use the **Zoom Back** and **Zoom Forward** (yellow arrow) buttons at the top of the activity graph to move backward and forward through previous zoom settings.

5   To scroll through the entire log, move the **Scroll chart** horizontal slider at the top left of the activity graph to move the view left or right.

IMPORTANT   Double-clicking on an individual line in the activity graph causes the corresponding entry in the log browser to be highlighted.

More About the Log Browser UI

The **vLogBrowser** pane displays formatted log entries. Each log entry is formatted into columns. For VMkernel logs, the column headers are: #, **Date**, **CPU**, **World ID**, and **Entry**. For hostd and other logs the column headers are: #, **Date**, **Source** and **Entry** (log text).

In the log browser, you can perform the following tasks:

■   Double-clicking an entry causes a dialog box to open, displaying other text associated with the log entry. Click **OK** when you are ready to dismiss the dialog box.

■   Filter for specific keywords by typing the keyword into the unlabeled text box on the top left of the Workbench Log Browser view and clicking **Filter**. The button is a toggle. If you wish to return to the unfiltered view of the log, click **Unfilter**.

■   Before clicking **Filter**, if you want to see the log entries immediately surrounding the entry with the keyword, select the number of surrounding entries to display from the numerical drop-down menu.

■   Sort the log entries by any column by clicking the column header once to sort in ascending order and click again to sort in descending order. Restore the original order by clicking the number ("#") column header to get ascending order.

■   Synchronize log entries in multiple log browsers by a specific timestamp. Select the **Sync** check box in each log browser that you want synchronized. Select a time stamped entry in one of the browsers. The other log browsers have the log entry closest to that timestamp selected.

■   Click the <**log_file_name.logx**> tab at the bottom of the **vLogBrowser** pane to see the .logx configuration file (XML formatted).

You can customize this log view by editing the XML in the .logx file. To see the changes, save, close and reload the file.

■   Various actions are available in the right-click menu, from the task bar icons, or from the **vLogBrowser** menu in the VMware Workbench menu bar:

a   **Filter** – The **Keyword Filtering with a Named Filter** dialog box displays.

In the **Choose a Filter Source** area, select one of the radio buttons: **This .logx file**, **Filters file** (and Browse for the file), or **Other open .logx files** (select the file from a drop-down list).

To create your own named filter, type a name in the **Named Filters** field or select an already existing named filter from the drop-down menu. Edit or accept the keywords for the filter in the regular expression formatting fields that follow. If you want to include lines surrounding the filtered lines, choose the number to include in the **Include # Surrounding Lines** selector.

To save your new named filter or edited filter, click Save.

To delete the filter in the Named Filters field, click Delete Filter. To clear the regular expressions form, click Clear Form.

To save the new or edited **Named Filter** and filter the `.logx` file, click Save and Filter. To unfilter the `.logx` file and clear the regular expression form, click Unfilter and Clear Form.

b   **Find** – The **Find by keyword** dialog box displays. Enter the keyword to search for in the **Keyword** text box. Click Find All to see all entries in the log browser containing the keyword selected. Click Hide to close the dialog box. Click Find next to find the next instance of the keyword in the log browser.

c   **Find Next** – When selected, takes you to the next entry containing the keyword you searched for previously. If you haven't selected keyword, it opens the **Find by keyword** dialog box.

d   **Find or Adjust Date**– The **Find by Timestamp or Adjust All Timestamps** dialog box displays.

To find a specific timestamp, type the desired timestamp in the **Date Input** field and click Find to go to the first entry in the log with that timestamp.

Or you can construct the timestamp using the calendar and clock controls in the dialog box. Changes you make using the calendar controls are reflected in the timestamp shown in the **Date Input** field. For example, changing the month to `February` changes the timestamp month to `02`. For the time portion of the timestamp, highlight a section of the clock display and click the buttons on the right to increment or decrement the number. For example, highlight the hour, and click the bottom button to decrement the hour number.

To adjust all of the timestamps in the log, set the desired timestamp in the Date Input field, and click Adjust. You can revert to the original timestamp by clicking Clear Adjustment.

e   **Highlight** – The **Highlight a keyword** dialog box displays. Type in the keyword to highlight and click Highlight. All instances of the keyword in the log are highlighted. Click Unhighlight to turn off the highlighting. Click Hide to close the dialog box.

To highlight keywords using the Highlight button on the Workbench Log Browser interface, type in a keyword in the unlabeled text box on the right side of the Workbench Log Browser view and click Highlight. The button is a toggle. If you wish to remove highlighting of the keyword, click Unhighlight.

f   **Copy –** Copy the selected entry or entries to the clipboard. You can open a text editor and paste the entries into a document.

g   **Refresh** – If additional log entries have been added to the log since you started analyzing it, you can select **Refresh** from the menu to bring those additional entries into the view. This works for logs on the local file system as well as logs from the ESXi hosts you are connected to.

h   **View Log Info** – A pop-up window displays the list of logs and their locations that you currently have loaded in this Workbench Log Browser project.

i   **Add Log Files** – Add local logs files to an existing Workbench Log Browser project (`.logx` file). The **Add local log files to table** dialog box displays. Browse to the log directory and select one or more logs to add. Click Add to add the files to the `.logx` file. The log display refreshes to include the new entries.

Log Browser Command Line

Workbench Log Browser packages include a command-line client (`vmwb–lbcli`) and server (`vmwb–server`) that allow you to operate on logs, log bundles, and recorded log sessions using the command-line. This appendix covers the following topics:

Command-Line Client Use Cases

This command-line client is meant to enable use cases such as the following:

- Using the log browser during headless automated testing.

  At whatever point is appropriate in your tests you can run a shell script that connects to a remote system, downloads certain types of logs and looks for patterns that indicate that failures or errors occurred.

- Creating scripts of repeated actions.

  If you need to do a repeated sequence of actions on a given type of log, for example once for each of a number of different hosts, you can create a script that performs those actions for you, such as connecting to the host, downloading certain types of logs, and applying a filter to that log.

- Using the log browser as a qualification tool for known defects.

  Log analysis is often the primary means of identifying defects. It would be helpful to quickly identify if a log bundle matches a pattern that corresponds to a known defect. The log browser command-line enables this sort of analysis. You can load the log bundle and then apply filters and search the log for matching entries.

- Doing manual log analysis from the command line

  If the system in which you are doing log analysis is a purely command-line system, it is convenient to do log analysis activities like loading a log bundle, applying filter and printing the results on the screen or to a file right from the command-line without needing to run any sort of GUI application.

Where to Find the Command-Line Client-Server

The Workbench Log Browser command-line client-server feature is available in the following packages:

-
-

Workbench Log Browser Update Site Plug-in for the Workbench Appliance

The command-line client and server are automatically installed when you download and install the Workbench Log Browser plug-in in the Workbench GUI (**Help > Install New Software**). When you install the update site, it installs the command-line only log browser server (`vmwb-server`) and the Log Browser command line client (`vmwb-lbcli`). The executables are automatically added to your path in the virtual machine.

Workbench Log Browser Native Eclipse IDE

The install packages for the native Eclipse version of the Log Browser, are self-contained platform-dependent ZIP files. Each platform specific package includes the command-line only log browser server (`vmwb-server`) and the log browser command-line client (`vmwb-lbcli`), and incudes a full working instance of a subset of the Workbench tools (including the Log Browser GUI) that can run on any machine.

The advantage of this package is that it can be used on any pre-existing host of your choosing. The packages are lightweight in the sense that they require no installation. Each package is a ready-to-use archive that you extract to your preferred directory.

To be able to launch the log browser server or command-line client from any shell, you must add them to the path.

> **CAUTION** Log Browser requires 64-bit Java for Mac and Linux and either 32-bit or 64-bit Java for Windows. The JVM is no longer packaged with the toolsuite. Download and install either a 32-bit (for Windows only) or 64-bit JVM for your platform from http://www.java.com/en/download/manual.jsp.
>
> If you are using the 32-bit Windows JVM, be sure to set JAVA_HOME and your PATH locally. If you set the JAVA_HOME variable globally, 64-bit-reliant packages will not find the correct JAVA package.

The following is a list of the available platform-dependent packages:

- rcp-toolsuite-lin64-3.5.0.*.zip
- rcp-toolsuite-mac64-3.5.0.*.zip
- rcp-toolsuite-win32-3.5.0.*.zip or rcp-toolsuite-win64-3.5.0.*.zip

Using the Command-Line with the Workbench GUI

To use the command-line client, you can either enable the server from the Workbench GUI (**VMware > Preferences**) or start the server from the command-line. If you want to move back and forth from the client command-line to the Workbench GUI, enable the server inside the GUI.

The advantage of using the command-line client in the context of the Workbench appliance is the seamless interoperability between the command line client and GUI operations. Logs that you open in the GUI will be available in the command line, and the reverse is also true, that is, logs opened in the client are available in the GUI. In the GUI also, you can record log browser sessions to create scripts (which are just a sequence of log browser commands).

The disadvantage of using the client with the Workbench GUI is that the Eclipse GUI must be running, thus it cannot be started from a shell. Another perceived disadvantage might be that the Workbench uses a fixed port number for the server. This means there can only be one server instance running on a given host. This limitation can be overcome with the standalone server.

Enable the server from the Workbench GUI

1 In the Workbench GUI, select **Windows > Preferences** to open the Preferences page.

2 In the Preferences page, expand **VMware** and select **Web Server** to display the Web Server options.

3 Select the **Enable web server on start up** check box, click **Apply**.

4 (Optional) To check if the web server has started, click **Check Status**.

5 Click **OK** to exit the Preferences page.

Using the Command-Line Client with the Standalone Server

The new command-line only server (`vmwb-server`) can be used to run the server manually from any shell. The server must be provided a path to a location to use for a workspace, that is a place to cache log files that are read during the process of log analysis. This workspace location must be unique to this server instance and cannot be shared with other server instances or with the IDE.

When you start the command-line server, it dynamically allocates ports that clients use for communication. The server records these ports in a file in the current users home directory so that when the vmwb-lbcli command-line client starts, it can read these values and know what ports to connect to. This functionality is seamless from your perspective as a user.

Start the command-line server from the command line

1 Open a terminal inside the Workbench virtual machine.

2 The server can be started in the background, for example:

```
$ vmwb-server /path/to/your/workspace &
```

The IDE currently is configured to use a fixed default port of 12443 for the server. You can start the server at the command in a mode where it also uses this same default port by doing:

```
$ vmwb-server /path/to/your/workspace --default-port &
```

Of course this mode means that you cannot start more than one instance of this server on a given host since it uses the same port number.

If you cannot have the server on the default port, you can manually specify which port(s) you would like it to use. There are actually two ports, one for the web services (12443 by default) and one for an administration interface (12221 by default). In the example that follows, the command is given with a value of 0 (which causes the dynamic allocation of the ports) but you can change this value to anything you wish.

```
$ vmwb-server /path/to/your/workspace --https-port=0 --admin-port=0 &
```

The command-line client application for the current user (the user that started the server) will automatically figure out the ports to use.

Permission Issues when Retrieving a Recorded Log Browser Session

By default on the Workbench virtual appliance, the IDE is run as `root` even though you may be logged in as a normal user (for example, the `vmware` user). One side effect of this is that files that are created by the IDE, such as the recorded shell script/batch file, are owned by `root` by default. If you log into Workbench as `root`, you will not have any permission issues in retrieving the recorded log browser session script. However, if you have logged into Workbench as a regular user, such as `vmware`, you will lack permissions to do anything with the script. To fix this, use one of the following commands to change the permissions:

■ Change ownership of the file.

```
sudo chown vmware:vmware lb-script.sh
```

■ Change permissions of the file so that user `vmware` can access it.

```
sudo chmod o+rwx lb-script.sh
```

Command-Line API Reference

This section describes the command-line client API and contains the following topics:

- "Command Syntax" on page 102
- "Getting the List of Available Commands" on page 102
- "Getting Help on a Particular Command" on page 102

Command Syntax

The command-line client, `vmwb-lbcli`, has syntax consistent with tools like `git` or `p4` from the command line, that is, you give the executable name, a command, command specific option, and then command specific arguments, if applicable.

There are many options that are common to various commands such as `--project`, which specifies the name of the project you are operating on, as well as `--resource` which is the resource within the project. Many of these commands can inherit the values for the command from the environment, if certain environment variables are defined. For example, if you do not specify the `--resource` option for a command that requires it, the value of the `VMLB_RESOURCE` environment variable, if defined, will be used. See the help for particular commands to see which variables are applicable if you desire to use this facility.

Getting the List of Available Commands

```
$ vmwb-lbcli help
vmwb_lbcli <command> [options]
    define    Define an additional user provided logx file type to the log browser server.
    save      Save the specified filter.
    count     Print the number of log entries currently filtered in.
    select    Select a log to be loaded in the log browser server.
    adjust    Adjust log entry timestamps by the specified time value.
    delete    Delete bundle and cached files for the project host.  Optionally specify
               particular resource.
    download  Download and analyze a log file from the given host.
    print     Print the specified log entries to the console.
    find      Find the index of log entries that match the specified pattern.
    filter    Appy the filter with the specified name.  The filter must have already been added.
    getfilter Get filters/a single filter if name specified.
    help      Display top level help information.
    version   Display program version number.
```

Getting Help on a Particular Command

For detailed help on a particular command, type the following:

```
vmwb-lbcli help <command>
```

Use Case Examples

The section lists some possible use cases and the client commands:

- "Printing Log Entries to the Console" on page 103
- "Downloading Log Files from a Remote Host for Analysis" on page 103
- "Discovering Logs on a Host" on page 104
- "Downloading All Logs for a Host" on page 104
- "Getting the Count of the Total Number of Visible Log Entries" on page 104
- "Printing Log Entries to stdout" on page 104
- "Applying a Simple Filter" on page 104
- "Exporting Entries to a File" on page 105

Using client commands, you can do the following tasks:

- Aggregate logs from multiple sources and combine them.

- Load a subset of log types from a log bundle.

- Count the number of log entries in a given log (unfiltered or filtered).

- Find log entries that match patterns.

- Print log entries to output files, and print in various formats (txt, xml, json, csv).

- Adjust time stamps in logs.

Printing Log Entries to the Console

Print the specified log entries to the console.

```
vmwb_lbcli print [options]
    --count <arg>     Optional max count of the number of entries to display.  Defaults to 30.
                      You may also specify 'all' in order to fetch all log entries.
    --first <arg>     Optional 1 based entry number of the first entry to display.
    --format <arg>    Optional format for the output.  Must be one of json,xml,txt.  Default is
                      txt.
    --lbserver <arg>  The host name of the log browser server to connect to.  If not provided
                      looks for env variable VMLB_SERVER, and then defaults to localhost.
    --noextra         If given, suppress printing of 'extra' lines in the output, leaving only
                      the first line of a given log entry as the value emitted.  Default is to
                      emit the extra lines.
    --nofilter        If given, the printed results are the UNFILTERED log entries.  By default
                      the entries printed are a result of the currently applied filter.
    --outfile <arg>   Redirect the output to the specified file instead of stdout.
    --project <arg>   The name of the project.  If not provided looks for env variable
                      VMLB_PROJECT.
    --resource <arg>  The name of the resource.  If not provided looks for env variable
                      VMLB_RESOURCE.
    --verbose         Verbose display of results if available.
```

Downloading Log Files from a Remote Host for Analysis

The download command allows you to connect to a remote host, collect a particular type of log file from it, and open it for analysis:

```
$ vmwb-lbcli download --host=192.168.1.106 --htype=esx --user=root --project=test --type=hostd
[Enter password for connection:]
```

Values for --htype can be the following:

Table A-1. Values for the --htype Option

vc=	Windows VC
vcva=	Linux VC
esx=	ESXi host
ssh=	SSH Server

:The command shares the workspace of the server it connects to, and by implication it also shares all projects, log analyses, and cached target credentials that the server has.

In this example, the user had not connected to this target yet, and because the command did not explicitly specify the password with the --password switch, it prompted the user for it.

For subsequent commands, the password does not have to be entered again because the credentials are cached in the servers secure storage. In the following example, a later download of a different log file type, it is not necessary to specify the user name and password. The cached ones from the connection above are used.

```
$ vmwb-lbcli download --host=192.168.1.106 --htype=esx  --project=test --type=VMkernel
$
```

After downloading the files, if you have the Workbench IDE running, you will see projects that have been created as a result of these commands. The projects include the server it connects to, as well as a new remote systems target for the host that was specified in the command-line.

Discovering Logs on a Host

If you do not know the types of logs that your remote system supports, the `download` command can give you some guidance. Omitting the `--type` argument causes the system to return an error message that lists the allowable types:

```
$ vmwb-lbcli download --project=myProj --host=192.168.1.106 --htype=esx --user=root
            --password=VMwar3rules
ERROR: Log type must be supplied.  Allowable values for this host include
            vmware,VMKernel,hostd,shell,syslog,vpxa,fdm,VMKwarning.
```

Downloading All Logs for a Host

The `download` command supports `--type=all` as a way to load all known log types for the specified host:

```
$ vmwb-lbcli download --project=myProj --host=192.168.1.106 --htype=esx --user=root
            --password=VMwar3rules --type=all
Loaded 8 log file types to project myProj:
    vmware                  192.168.1.106_vmware.logx
    VMKernel                192.168.1.106_VMKernel.logx
    hostd                   192.168.1.106_hostd.logx
    shell                   192.168.1.106_shell.logx
    syslog                  192.168.1.106_syslog.logx
    vpxa                    192.168.1.106_vpxa.logx
    fdm                     192.168.1.106_fdm.logx
    VMKwarning              192.168.1.106_VMKwarning.logx
```

Getting the Count of the Total Number of Visible Log Entries

The `count` command gives you the count of the number of log entries that are visible with the current filter applied (if there is one). There is an additional `--nofilter` command line option that will give you the unfiltered count if you wish to get that without removing the filter. In the example that follows, the number that shows below the command is the answer returned by the system.

```
$ vmwb-lbcli count --project=test --resource=192.168.1.106_hostd.logx
471686
```

Printing Log Entries to stdout

The `print` command, used to print log entries, is flexible in terms of the format of the reporting (`txt`, `xml`, `json`), as well as the number of entries it prints and the starting entry. You must specify at least the project and resource within the project as arguments to the command. Other arguments, such as formatting, are optional.

The following is an example of the default behavior, which is to print the first 30 entries matched by the currently applied filter, if one exists. In this example, there is no filter applied.

```
$ vmwb-lbcli print --project=test --resource=192.168.1.106_hostd.logx
1 |   |   | Section for VMware ESX, pid=2853, version=5.0.0, build=build-623860, option=Release |
            ------ In-memory logs start --------
2 | 2012-07-13 03:23:48.100 -0600 | FFE2BAC0 info 'Default' | Supported VMs 63 |
<snip...>
30 | 2012-07-13 03:23:48.104 -0600 | FFE2BAC0 info 'Default' | Trying hbrsvc |
```

Applying a Simple Filter

The `filter` command supports two kinds of filters: defining a simple filter dynamically with the `--pattern` option (a `regex` pattern), or applying a known named filter. In the following example, a simple filter is applied, the count of filtered entries is retrieved, and the entries are printed in XML format:

```
 $ vmwb-lbcli filter --project=test --resource=192.168.1.106_hostd.logx --pattern="Event 157"
```

```
$ vmwb-lbcli count --project=test --resource=192.168.1.106_hostd.logx
2


$ vmwb-lbcli print --project=test --resource=192.168.1.106_hostd.logx --format=xml
<logItems>
  <logItem>
    <_>411049</_>
    <Date>2012-08-03 16:25:32.215 -0600</Date>
    <Source>47B01B90 info 'ha-eventmgr'</Source>
    <Entry>Event 157 : User root logged out</Entry>
    <EntryExtraLines/>
  </logItem>
  <logItem>
    <_>471663</_>
    <Date>2012-08-03 16:25:32.215 -0600</Date>
    <Source>47B01B90 info 'ha-eventmgr'</Source>
    <Entry>Event 157 : User root logged out</Entry>
    <EntryExtraLines/>
  </logItem>
</logItems>
```

Exporting Entries to a File

The `print` command is used to export entries. This can be done many different ways depending on your needs. Several examples are given.

stdout Redirection

The normal `print` command directs its output to `stdout`. You can redirect this to a file. This particular example also demonstrates the use of the `--count` and `--format` options:

```
$ vmwb-lbcli print --project=test --resource=192.168.1.106_hostd.logx --count=1 --format=json >
            myOutput.json
$ cat myOutput.json
{
    "logItems": [
        {
            "#": "411049",
            "Date": "2012-08-03 16:25:32.215 -0600",
            "Source": "47B01B90 info 'ha-eventmgr'",
            "Entry": "Event 157 : User root logged out",
            "EntryExtraLines": ""
        }
    ]
}
```

The --outfile Option

The `print` command also has an `--outfile` option for directly specifying the output file. This option has the advantage of being able to display a percentage complete status in the console as the download of the entries progresses. This may be of interest in the case of very large log files. The following example downloads a large number of log entries. (The `--count=all` option was used to specify all log entries.)

NOTE In this example, the filter was first removed with the `filter` command before printing. The `--nofilter` option on the `print` command could also have been used to do this without removing the filter.

```
$ vmwb-lbcli filter --project=test --resource=192.168.1.106_hostd.logx --remove
$ vmwb-lbcli print --project=test --resource=192.168.1.106_hostd.logx --format=xml --count=all
              --outfile=myFile.xml
Downloading log entries: 100% complete
Done writing to file /home/aspear/myFile.xml
downloaded 471686 entries in 62.909s (7497 entries/sec)
```

Finding Particular Entries and Printing Them

As an alternative to filtering, you can also use the `find` command to search for particular patterns. The `find` command by default returns the log entry number of the matching entries:

```
$ vmwb-lbcli find --project=test --resource=192.168.1.106_hostd.logx --pattern="Event 157"
411049
471663
```

You can print the entries in any given format using the `--print` option:

```
$ vmwb-lbcli find --project=test --resource=192.168.1.106_hostd.logx --pattern="Event 157"
            --print=txt
411049 | 2012-08-03 16:25:32.215 -0600 | 47B01B90 info 'ha-eventmgr' | Event 157 : User root
            logged out |
471663 | 2012-08-03 16:25:32.215 -0600 | 47B01B90 info 'ha-eventmgr' | Event 157 : User root
            logged out |
```

Finding Particular Entries and Failing if the Count is Not What You Expect

Both the `find` and `count` commands support a couple of additional options that can be useful for use cases such as automated regression testing:

```
--fail_if_count_equal <arg>      Optional integer argument to check resulting line count against.
                                 If the count is equal to this number the command fails.
                                 Mutually exclusive with the fail_if_count_not_equal option.
--fail_if_count_not_equal <arg>  Optional integer argument to check resulting line count against.
                                 If the count is NOT equal to this number the command fails.
                                 Mutually exclusive with the fail_if_count_equal option.
```

In the following example, `Failure` is used in a particular log file in the case of any error, and there are 17 instances of `Failure` that are known issues and always emitted in the log. As a part of an automated regression test, this log file could be checked and would fail if the count is more than 17 (failing in this case means that the `vmwb-lbcli` process returns an exit code of non-zero in addition to printing an error message to `stderr`).

```
$ vmwb-lbcli find --project=test --resource=192.168.1.106_hostd.logx --pattern="Failure"
            --fail_if_count_not_equal=17
1162
6916
(... 17 entry line numbers printed here)
464835
ERROR: Count 18 is not equal to fail_if_count_not_equal value 17.
```

Printing the Contents of a Local Log Bundle File

The `bundle` command allows printing of information on the contents of a log bundle, as well as loading all or a subset of the contained log file types in the bundle.

The following example prints the available log types in a bundle:

```
$ vmwb-lbcli bundle /home/aspear/logfiles/gss/vcsupport-2011-10-26.16479.zip
Examining log bundle /home/aspear/logfiles/gss/vcsupport-2011-10- 26.16479.zip ...
Available log types:
    lookup Server
    imsTrace
    eam
    stats
    vpxd
    jointool
    ssoAdminServer
    sso-service-cfg
    SRM Recovery
    Virgo Server
    install
    cim-dia
    SRM Protected
    sms
```

Loading Log Types from a Log Bundle

You can load log types from a bundle by adding one or more ––load options to the bundle command. The load options specify the log types you wish to load. The command requires that you must specify a project name that you wish to load the given log types into. The following is an example of loading a couple of log types:

```
$ vmwb-lbcli bundle --project=test2 --load=vpxd --load=stats
             ~/logfiles/gss/vcsupport-2011-10-26.16479.zip
Loading log bundle /home/aspear/logfiles/gss/vcsupport-2011-10-26.16479.zip to project test2 ...
Loaded 2 log file types:
    stats                 Local_stats_vmsp.logx
    vpxd                  Local_vpxd_vmsp.logx
```

You can also specify ––load=all to load all types in the bundle:

```
$ vmwb-lbcli bundle --project=test2 --load=all ~/logfiles/gss/vcsupport-2011-10-26.16479.zip
Loading log bundle /home/aspear/logfiles/gss/vcsupport-2011-10-26.16479.zip to project test2 ...
Loaded 14 log file types:
    lookup Server         Local_lookup_Server_vmsp.logx
    imsTrace              Local_imsTrace_vmsp.logx
    eam                   Local_eam_vmsp.logx
    stats                 Local_stats_vmsp.logx
    vpxd                  Local_vpxd_vmsp.logx
    jointool              Local_jointool_vmsp.logx
    ssoAdminServer        Local_ssoAdminServer_vmsp.logx
    sso-service-cfg       Local_sso-service-cfg_vmsp.logx
    SRM Recovery          Local_SRM_Recovery_vmsp.logx
    Virgo Server          Local_Virgo_Server_vmsp.logx
    install               Local_install_vmsp.logx
    cim-dia               Local_cim-dia_vmsp.logx
    SRM Protected         Local_SRM_Protected_vmsp.logx
    sms                   Local_sms_vmsp.logx
```

After the bundle is loaded, normal operations can be done against the file, such as finding, printing, and filtering, by referencing the project and resource names (printed in the previous example):

```
$ vmwb-lbcli print --project=test2 --resource=Local_vpxd_vmsp.logx --count=1 --first=10
10 | 2011-10-25 23:46:56.666 -0600 | EAFFF700 info 'ThreadPool' | Thread enlisted |
```

Known issues/Limitations

The log browser server must be running in order for the command to work.

Currently only local log bundles can be loaded. There is no facility to download a remote a log bundle from a Windows host, and also no integration to request a host to create a log bundle from the CLI. If you can copy the Windows log bundles to your localhost, then they can be analyzed.

Recordings don't actually record the content of filters that are defined. They record the name of the filter and the activation of filters, but rely on the filter being persisted on the server side.

Advanced Use Cases

It is possible to connect to a log browser server running on any machine you wish. By default the assumption is that the server is connected to localhost, but the ––lbserver option can be used to connect to a server on a different host. Related to this, you can also specify the port to connect on as a part of the host name, for example ––lbserver="myServer.eng.vmware.com:12444". Workbench Web Services are fixed on port 12443.

Index