Retail passion. Innovative solutions.

Retek® Point-of-Sale™ 11.0

Mission Control User Guide

Corporate Headquarters:

Retek Inc.
Retek on the Mall
950 Nicollet Mall
Minneapolis, MN 55403
USA

888.61.RETEK (toll free US)
Switchboard:
+1 612 587 5000

Fax:
+1 612 587 5100

European Headquarters:

Retek
110 Wigmore Street
London
W1U 3RW
United Kingdom

Switchboard:
+44 (0)20 7563 4600

Sales Enquiries:
+44 (0)20 7563 46 46

Fax:
+44 (0)20 7563 46 10

Printed in the United States of America.

Customer Support

Customer Support hours

Customer Support is available 7x24x365 via email, phone, and Web access.

Depending on the Support option chosen by a particular client (Standard, Plus, or Premium), the times that certain services are delivered may be restricted. Severity 1 (Critical) issues are addressed on a 7x24 basis and receive continuous attention until resolved, for all clients on active maintenance. Retek customers on active maintenance agreements may contact a global Customer Support representative in accordance with contract terms in one of the following ways.

Contact Method	Contact Information
E-mail	support@retek.com
Internet (ROCS)	rocs.retek.com Retek's secure client Web site to update and view issues
Phone	+1 612 587 5800

Toll free alternatives are also available in various regions of the world:

Australia	+1 800 555 923 (AU-Telstra) or +1 800 000 562 (AU-Optus)
France	0800 90 91 66
Hong Kong	800 96 4262
Korea	00 308 13 1342
United Kingdom	0800 917 2863
United States	+1 800 61 RETEK or 800 617 3835
Mail	Retek Customer Support Retek on the Mall 950 Nicollet Mall Minneapolis, MN 55403

When contacting Customer Support, please provide:

- Product version and program/module name.
- Functional and technical description of the problem (include business impact).
- Detailed step-by-step instructions to recreate.
- Exact error message received.
- Screen shots of each step you take.

Contents

Chapter 1 – Introduction

The Retek ® RPOS™ Tools Mission Control is a Java®-based management application tool that allows you to manage the Retek® Point-of-Service™ (RPOS) application in a physically distributed system. With Mission Control, you can monitor, manage, debug, and tune multiple network devices simultaneously. Mission Control*'s* intuitive and user-friendly graphical-user interface (GUI) enables you to employ a simple point-and-click method to manage an entire network system.

The following are features and benefits of Mission Control:

- **Identify problems quickly**—Mission Control red flags exceptions and eliminates the need for you to page through lengthy log files. This allows you to identify problems and initiate a resolution quickly.
- **Optimize performance**—By balancing the use of multiple components, you can tune the application server for optimal performance based upon your business rules.
- **Extend component lifecycle**—You can quickly and easily allocate resources across a system to avoid overloads and extend the life of your components. Upgrades can be made dynamically, eliminating the need for downtime during system upgrades.
- **Point and click debugging**—Mission Control*'s* GUI enables you to view server statistics and pinpoint solutions by pointing and clicking.
- **Monitor system vitals**—Mission Control provides you with a window into a physically distributed system. This application tool enables you to proactively monitor critical system vitals such as hits per period, HEAP, CPU utilization, and maximum number of concurrent requests. These system vitals can be graphed to easily view and monitor performance trends.
- **Send messages and view network topology**—The Message Monitor feature of Mission Control allows you to publish messages in a distributed computing environment.
- **Automated update process**—Mission Control allows you to update application servers and clients using an automated process called a manifest.

Logon to mission control

Before you can begin using Mission Control, you will be prompted for a password while logging on. This extra security feature helps ensure only designated personnel can monitor and manage the RPOS system.

To log on to Mission Control:

1. Click the Mission Control icon on the desktop.
   Or
   If you are using a command prompt:
   a. Type **cd:** and then the path to the bin folder.
   b. Type **javaenv** to start the Java environment.
   c. Type **mc**.

   The Logon to RPOS Tools Mission Control dialog box is displayed.

2. On the **Logon to RPOS Tools Mission Control** dialog box, type the password in the **Enter Password** field.

Note: The default password is **retek**. It is recommended that you change the password to a user-defined password after you log on to Mission Control for the first time.

Note: If you type an incorrect password, the **Incorrect Password** dialog box is displayed. To close the dialog box, click **OK**. Retry typing the password.

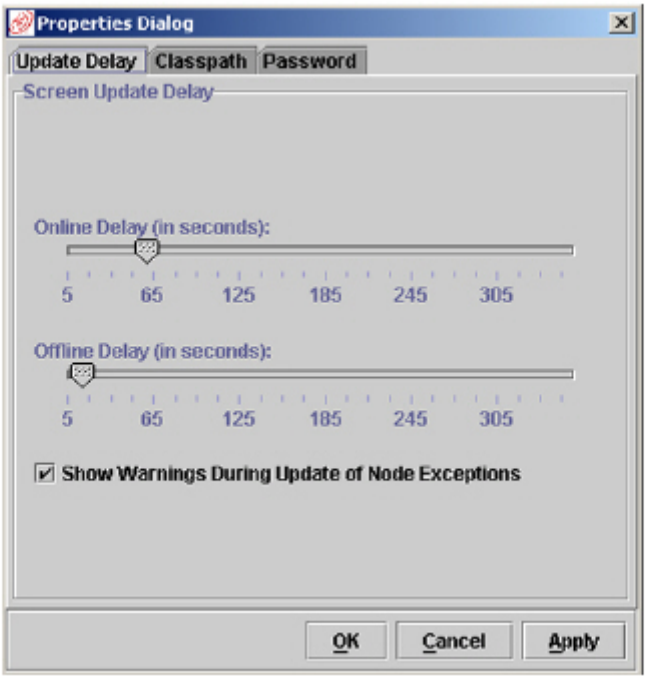
3. Click the **Login** button. Mission Control is started and displayed on the screen.

Change your logon password

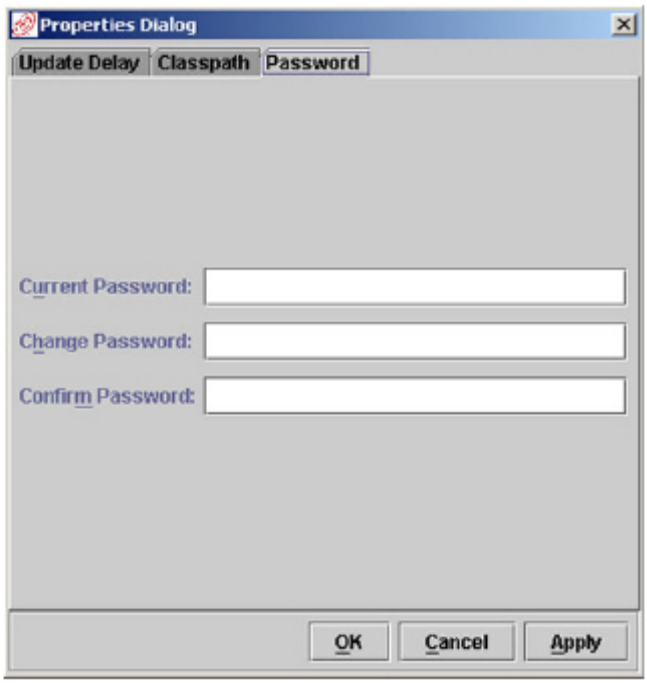
It is recommended you change your password after logging on to Mission Control for the first time. Currently there are no parameters or restrictions to a password. A password can be any combination of letters, numerals, spaces, and symbols. The number of characters is user-defined.

To change your logon password:

1. Select **Options>Properties**. The **Properties** dialog box is displayed. The **Update Delay** tab is the default tab.



2. Click the **Password** tab.



Note: To navigate through fields, select TAB or ENTER, or select the field in which you wish to enter the password information.

3. In the **Current Password** field, type the current password.
4. In the **Change Password** field, type the new password.
5. In the **Confirm Password** field, re-type the new password.

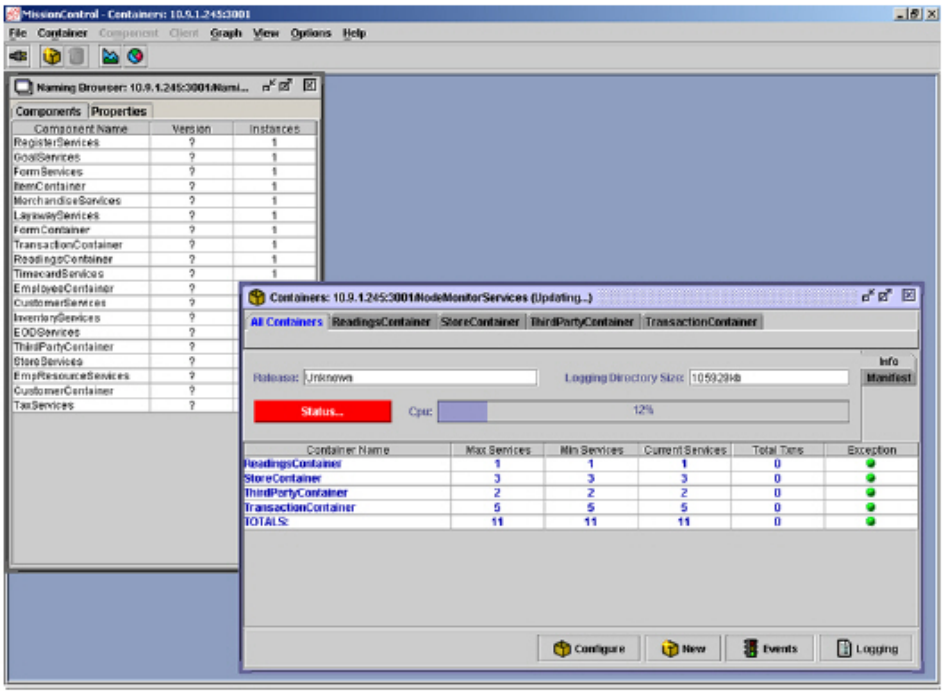
Note: If you re-type the password incorrectly, the system displays an error message. Select **OK**, and re-type the password correctly.

6. To change the password, click **OK**. The **Properties** dialog box is no longer displayed.
   Or
   Select the **Apply** button. The system is updated with the new password. The **Properties** dialog box is still displayed.

Mission control workspace overview

The **Mission Control** workspace consists of a single window containing the following:

- Menu bar
- Toolbar
- Windows
- Tabs



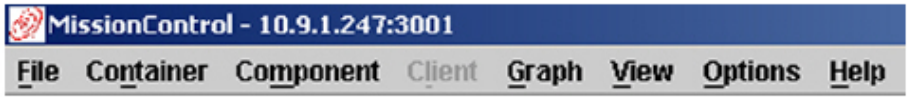
Mission Control workspace

Menu bar and menus

The menu bar contains the different menu options that you can choose from. Mission Control provides the following menu options:

- File
- Container
- Component
- Client
- Graph
- View
- Options
- Help

Note: When starting Mission Control, all menus are active except the **Containers**, **Components**, and **Clients** menus. You must first connect to and view a node before these menus become active.

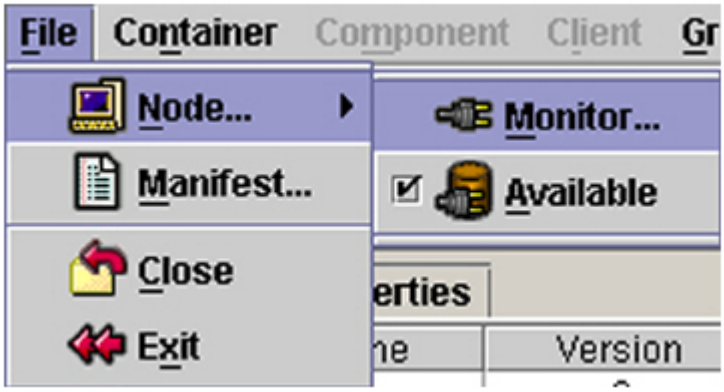


Mission Control menu bar

File Menu

The **File** menu provides options to exit Mission Control, close windows, and select and view nodes that were previously connected to during a session of Mission Control. The following are menu options on the **File** menu:

- **Node**—Allows you to connect to a node (Monitor*) or make all components or services available or unavailable on a node (Available).
- **Manifest**—Allows you to create, edit, and remove a manifest.
- **Close**—Allows you to close the active window.
- **Exit**—Allows you to close all active windows and exit Mission Control.



File menu and menu options

Note: Menu options marked with an asterisk (*) have corresponding command buttons available on the toolbar.

Container menu

The **Container** menu allows you to monitor and manage the containers running on a server. This menu also allows you to view log files for each container. The following are menu options on the **Container** menu:

Note: The **Container** menu and menu options are not active until you connect to an application server and select a container.

- **Configure**—Allows you to configure the properties of an existing container.
- **New***—Allows you to create a new container.
- **Delete**—Allows you to delete an existing container.
- **Start**—Allows you to start a new instance of a container.
- **Shutdown**—Allows you to stop or shutdown an instance of a container.
- **View Logs**—Allows you to view the log files of a container.

Note: An ellipsis (...) following a menu option signifies a new window will open if you select that option.

Note: Menu options marked with an asterisk (*) have corresponding command buttons available on the toolbar.

Container menu and menu options

Component menu

The **Component** menu allows you to monitor and manage the components running on a server. This menu also allows you to view log files for each instance of a component. The following are menu options on the **Component** menu:

Note: The **Component** menu and menu options are not active until you connect to a node and select a component.

- Configure—Allows you to configure the properties of an existing component.
- **New***—Allows you to create a new component.
- **Delete**—Allows you to delete an existing component.
- **Start**—Allows you to start a new instance of a component.
- **Shutdown**—Allows you to stop or shutdown an instance of a component.
- **View Logs**—Allows you to view the log files of a component.

Note: An ellipsis (...) following a menu option signifies a new window will open if you select that option.

Note: Menu options marked with an asterisk (*) have corresponding command buttons available on the toolbar.

Component menu and menu options

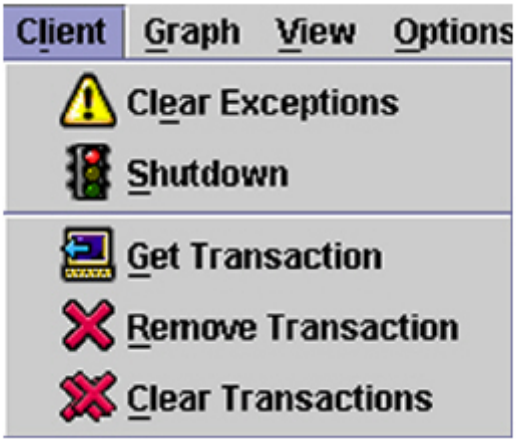
Client menu

The **Client** menu allows you to manager clients on the network from a remote location. The following are menu options on the **Client** menu:

Note: The **Client** menu and menu options are not active until you connect to a client.

- Get Exceptions—Allows you to retrieve exceptions.
- Shutdown—Allows you to shutdown processes on the selected client.
- Get Transaction—Allows you to retrieve broken transactions.
- Remove Transaction—Allows you to remove broken transactions on the selected client.
- Clear Transaction—Allows you to clear broken transactions on the selected client.

Note: An ellipsis (...) following a menu option signifies a new window will open if you select that option.



Client menu and menu options

Graph menu

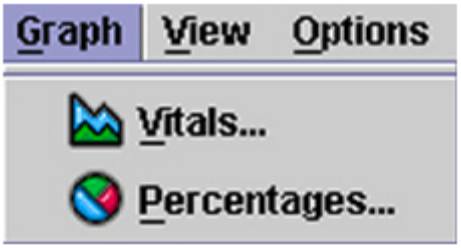
Note: The **Graph** menu and menu options are not active until you connect to a node.

The **Graph** menu allows you to collect and view system and server vitals in various graphical formats. The following are menu options on the **Graph** menu:

- **Vitals***—Allows you to view system and server vitals in configurable graphical formats.
- **Percentages**—Allows you to view server transaction percentages by service in configurable pie-chart format.

Note: An ellipsis (...) following a menu option signifies a new window will open if you select that option.

Note: Menu options marked with an asterisk (*) have corresponding command buttons available on the toolbar.



Graph menu and menu options

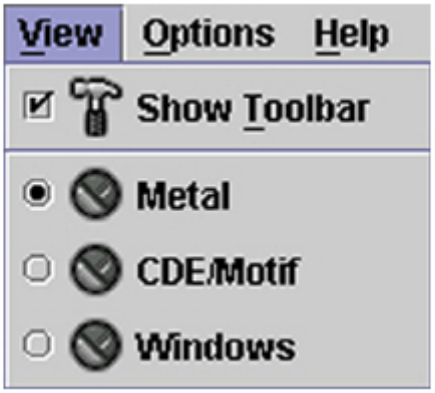
View menu

The **View** menu allows you to view the toolbar and change the look and feel of Mission Control. The following menu options on the **View** menu:

- **Show Toolbar**—Allows you to hide the toolbar or make the toolbar visible.
- **Metal**—A theme that changes the appearance of Mission Control.
- **CDE/Motif**—A theme that changes the appearance of Mission Control.
- **Windows**—A theme that changes the appearance of Mission Control.

Note: An ellipsis (...) following a menu option signifies a new window will open if you select that option.

Note: A check or bullet will appear by the selected menu option.



View menu and menu options

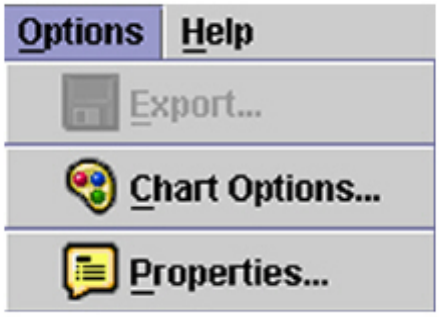
Options menu

The **Options** menu allows you to set server properties, register for email notifications, export system and performance information into a Microsoft® Excel® file, and adjust the chart appearance. The following are menu option on the **Options** menu:

- **Export**—Allows you to export system and performance information into an Excel spreadsheet.
- **Chart Options**—Allows you to customize the look and feel of a table.
- **Properties**—Allows you to update the window update delay, change, add, or remove a classpath, and change the logon password for Mission Control.

Note: An ellipsis (...) following a menu option signifies a new window will open if you select that option.

Note: Some menu options will appear inactive depending on the task you are performing.



Options menu and menu options

Help menu

The **Help** menu allows you to access the online help files for Mission Control. The help files will launch in a separate window.

Note: An ellipsis (...) following a menu option signifies a new window will open if you select that option.

Help menu and menu options

Toolbar and command buttons

Mission Control command buttons correspond with menu options. The command buttons on the toolbar will become active depending on the task being performed.

Note: See the section in this guide specific to the task you are performing for a list of available options.

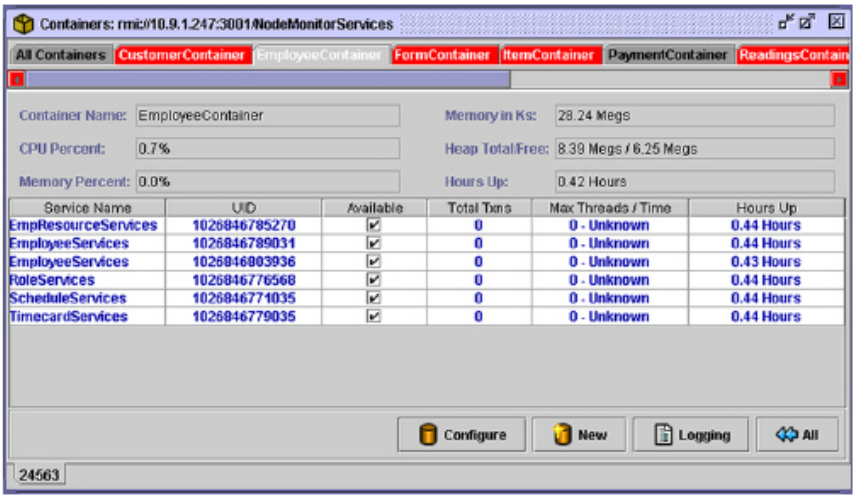
The table below lists and describes each command button that appears on the **Mission Control** toolbar.

Mission Control Command buttons		
Command Button	**Description**	**Menu**
Monitor Server 	Allows you to connect to and monitor registered nodes.	File
New Container 	Allows you to add a new container to an existing server.	Container
New Component 	Allows you to add a new component to an existing server.	Component
Vitals Graph 	Allows you to view system and server vitals in configurable graphical formats.	Graph
Percentages Chart 	Allows you to view server transaction percentages by service in configurable pie-chart format.	Graph

Note: To view a command button's definition, hover the cursor over the button.

Windows

Mission Control's windows are the main working area of the application. Selecting various menu options and command buttons opens a new working window.



An example of a Mission Control window

Opening a window makes that window the active window. Selecting another open window will change the active window. Also, each window can be minimized, maximized, and resized to your specifications. Placing the cursor on the window corner allows you to resize the window to fit your working preferences.

Tabs

Tabs allow you to navigate within windows. Tabs vary depending on the task being performed. Some tabs are not active when performing some tasks.

An example of tabs on a window

Chapter 2 – Set up a node

You must first register the node with Retek® RPOS™ Mission Control before you can begin viewing, monitoring, managing, debugging, or tuning your network system (for example, an application server, database server, or client residing in a store). You must provide the IP address and the port number of the node you wish to register. You only have to register a node once with Mission Control. After you register a node, you can connect to a node using the **Monitor Node** feature.

Register a node

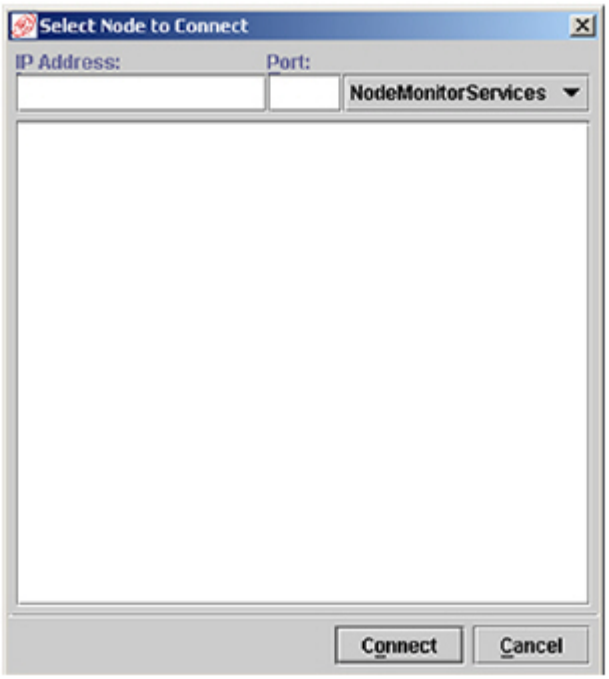
Registering a node allows you to connect to the following nodes or server-side components:

- Application server
- Client
- Naming server
- Message Monitor

To register a node:

1. Select **File>Node>Monitor**.
   Or
   Click the **Monitor Node** button.

The **Select Node to Connect** dialog box is displayed.



Note: To navigate through fields, select TAB or select the field in which you wish to type the node or server-side component information.

2. Type the IP address of the node you wish to register.
3. Type the port number of the node you wish to register. The following are default port numbers:
   - NodeMonitorServer (application server)—**3001**
   - IClientManager (client)—**2048**
   - MessageMonitor (Message Monitor)—**3000**
   - NamingServer (naming server)—**3000** (root naming server) or 3001 (specific node)
4. In the drop-down list, select one of the following:
   - To register an application server, select **NodeMonitorService**.
   - To register a client, select **IClientManage**.
   - To register a naming server, select **NamingServer**.
   - To register the message monitor, select **MessageMonitor**.

Note: NodeMonitorService is the default value.

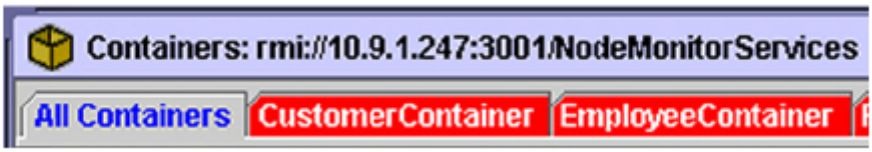
5. Click the **Connect** button. The appropriate window and information for the connected node is displayed
6. Repeat these steps to register additional nodes.

After you register a node, Mission Control assigns the node a name based on the information you provided. The following is the naming convention for all registered nodes: node's IP address: port number/type of node (for example, **NodeMonitorServices** for servers and **IClientManage** for clients).

The following is an example of a registered application server and a registered client:

- Registered application server—10.212.128.245:3001/NodeMonitorServices
- Registered client—10.212.125.225:2048/IClientManage

Note: There are no spaces in a node name.



Node name example

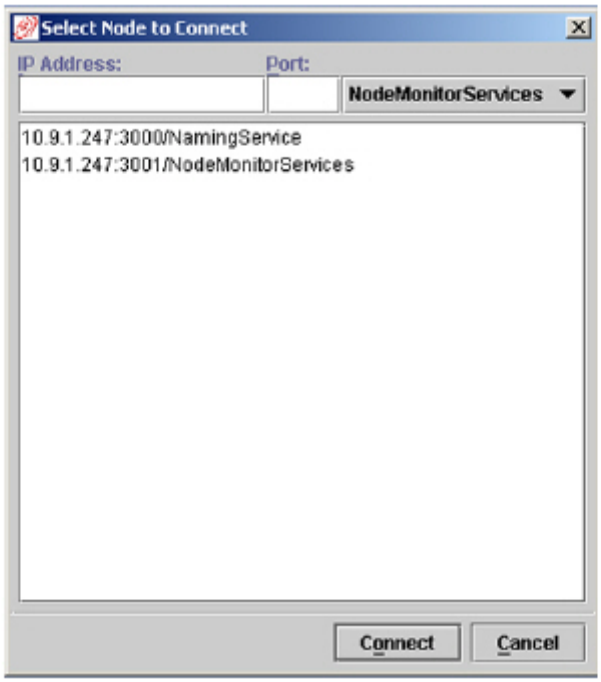
Apply an alias to a node

While Mission Control assigns a name to a registered node, you can apply an alias to the node. The alias does not replace the name assigned by Mission Control, but it is a managing tool for you to help keep track of multiple nodes. For example, you can apply an alias to a client based on the location of the store, or you can apply an alias to a application server based on where it is located within the United States.

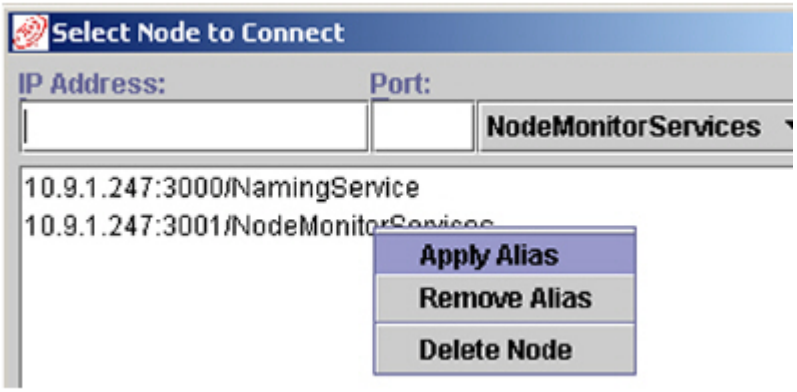
To apply an alias to a node:

1. Select **File>Node>Monitor**.
   OR
   Click the **Monitor Node** button.

The **Select Node to Connect** dialog box is displayed.

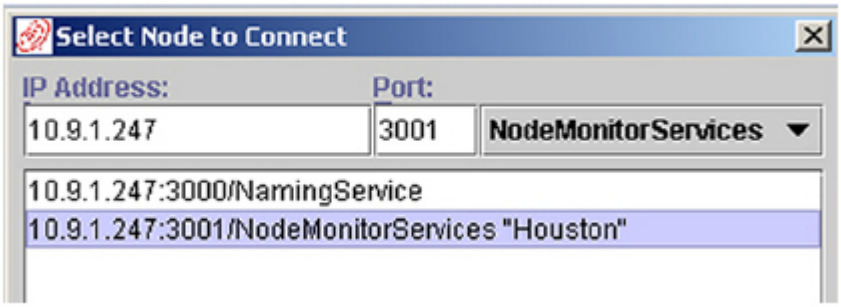


2. Select the node you wish to apply an alias to. The node is highlighted in blue.
3. Right click on the node. A pop-up list is displayed.



4. In the pop-up list , select **Apply Alias**. The **Apply Alias** dialog box is displayed.

5. In the **Apply Alias** dialog box, type an alias.
6. To apply the alias to the selected node, click **OK**.
   OR
   Click the **Enter** button. In the **Select Node to Connect** dialog box, the alias is displayed on the right-hand side of the node name.



Note: An alias only appears in the **Select Node to Connect** dialog box. After connecting to a node, the various Mission Control windows display only the node's name.

Change the alias of a node

Mission Control allows you to change a node's alias. The steps to changing are exactly the same as applying an alias. For more information on changing a node's alias, see "Applying an Alias to a Node".

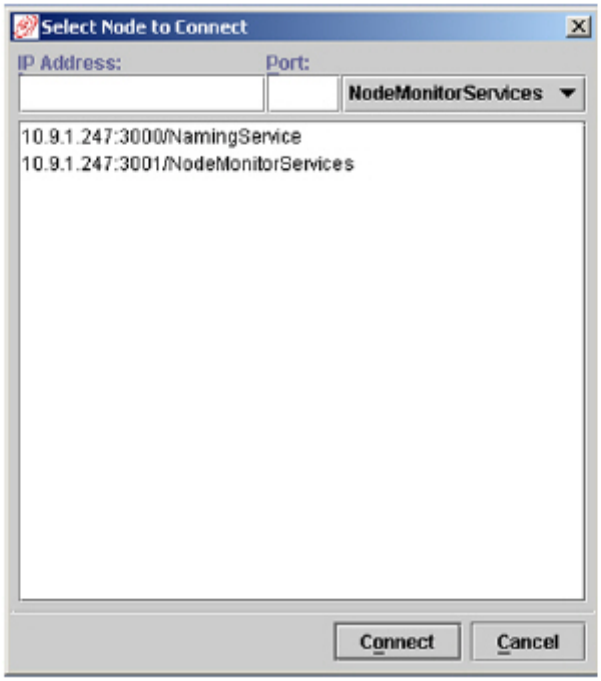
Remove an alias from a node

Note: Removing an alias only removes the alias from the node , not the node name assigned by Mission Control.

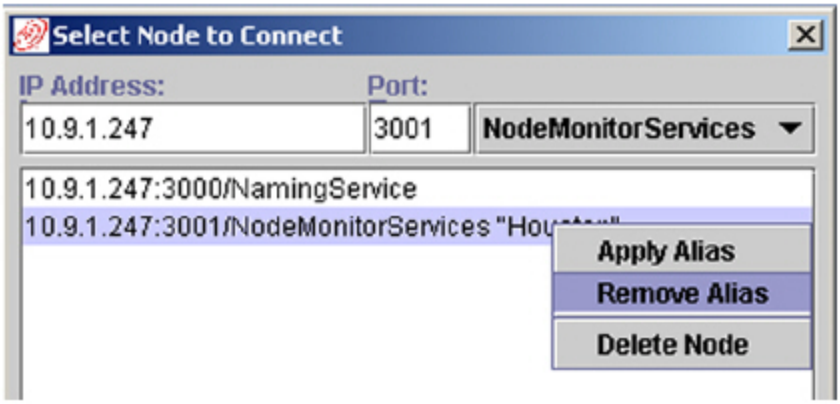
To remove an alias from a node:

1. Select **File>Node>Monitor**.
   OR
   Click the **Monitor Node** button.

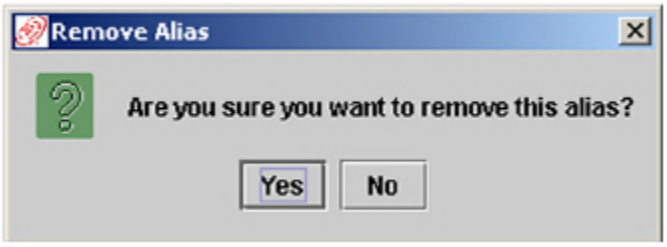
The **Select Node to Connect** dialog box is displayed.



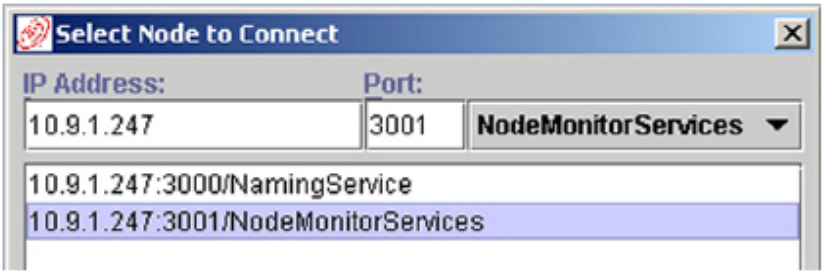
2. Select the node you wish to remove the alias from. The node is highlighted in blue.
3. Right click on the node. A pop-up list is displayed.



4. In the pop-up list, select **Remove Alias**. The **Remove Alias** dialog box is displayed.



5. In the **Remove Alias** dialog box, click **Yes**. The alias is no longer displayed in the **Select Node to Connect** dialog box.



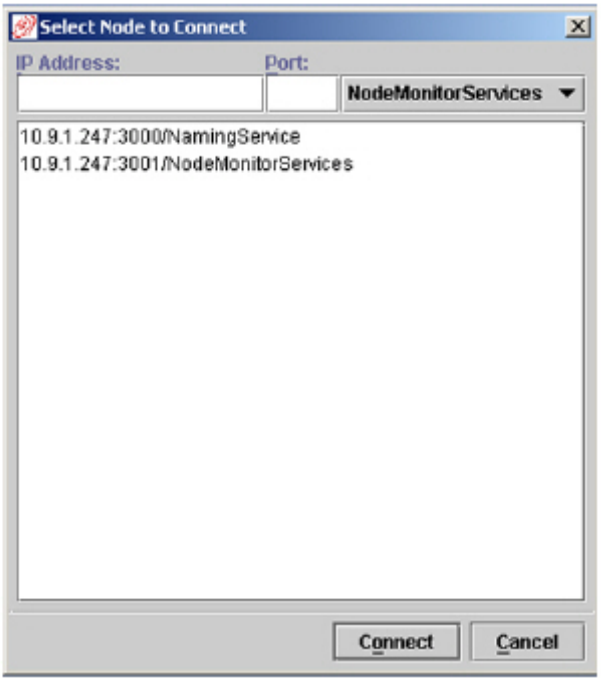
Connect to a registered node

After you have registered a node, Mission Control stores that node information. The **Select Node To Monitor** dialog box displays all the nodes that have been connected to using Mission Control.

To connect to a registered node:

1. Select **Node>Monitor**.
   OR
   Click the **Monitor Node** button.

The **Select Node to Monitor** dialog box is displayed.



2. In the **Select Node to Monitor** dialog box, select the node you wish to connect to. The selected node is highlighted in blue.
3. Click the **Connect** button. The appropriate window and node information is displayed.
4. To connect additional nodes, repeat steps 1-3.

Remove a registered node

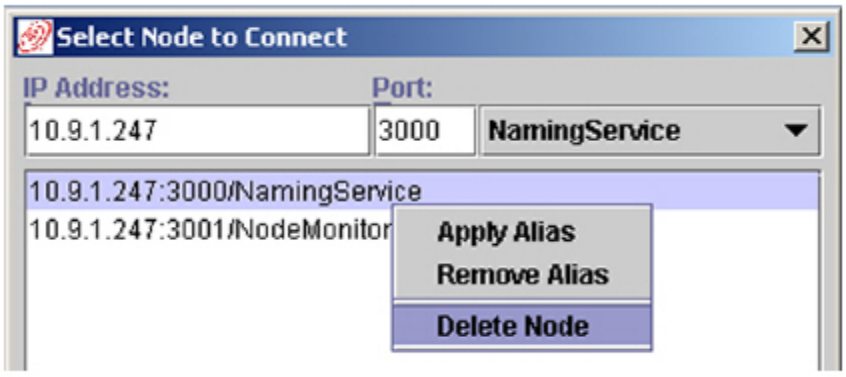
If you no longer need to connect to a particular node or if the node no long exists, you can remove the node from the **Select Node to Monitor** dialog box.

To remove a node from the **Select Node to Monitor** dialog box:

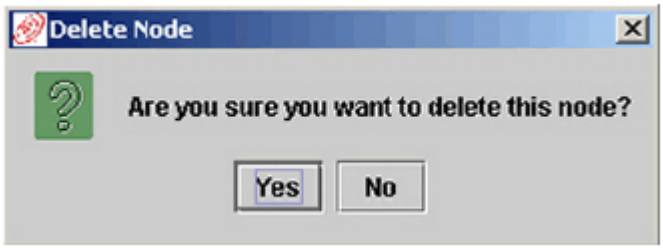
1. Select **File>Node>Monitor**.
   OR
   Click the **Monitor Node** button.

The **Select Node to Connect** dialog box is displayed.

2. Select the node you wish to delete. The node is highlighted in blue.
3. Right click on the node. A pop-up list is displayed.



4. In the pop-up list, select **Delete Node**. The **Delete Node** dialog box is displayed.



5. In the **Delete Node** dialog box, click **Yes**. The node is no longer displayed in the **Select Node to Connect** dialog box.

Chapter 3 – Working with containers

After connecting to an application server, you can view and monitor the processes running on a server. The Retek® Point-of-Service™ (RPOS) platform has individual components or services stored in containers, and the processes are run from the containers. A container can store multiple components. Containers reduce the memory load by combining multiple components into one process so the components share the memory. Containers also increase the horizontal scalability of the RPOS system.

You can perform the following tasks when working with containers:

- Configure an existing container
- Create a new container
- Start and stop an instance of a container
- Delete a container
- View container log files

Containers and components are configurable per the customer.

Containers and components	
Containers	**Components**
CustomerContainer	• CustomerServices
EmployeeContainer	• EmpResourceServices • EmployeeServices • RoleServices • ScheduleServices • TimecardServices
FormContainer	• FormServices
ItemContainer	• InventoryServices • ItemServices • MerchandiseServices • TransferServices
PaymentContainer	• RedeemServices
ReadingsContainer	• ReadingServices
StoreContainer	• GoalServices • RegisterServices • StoreServices

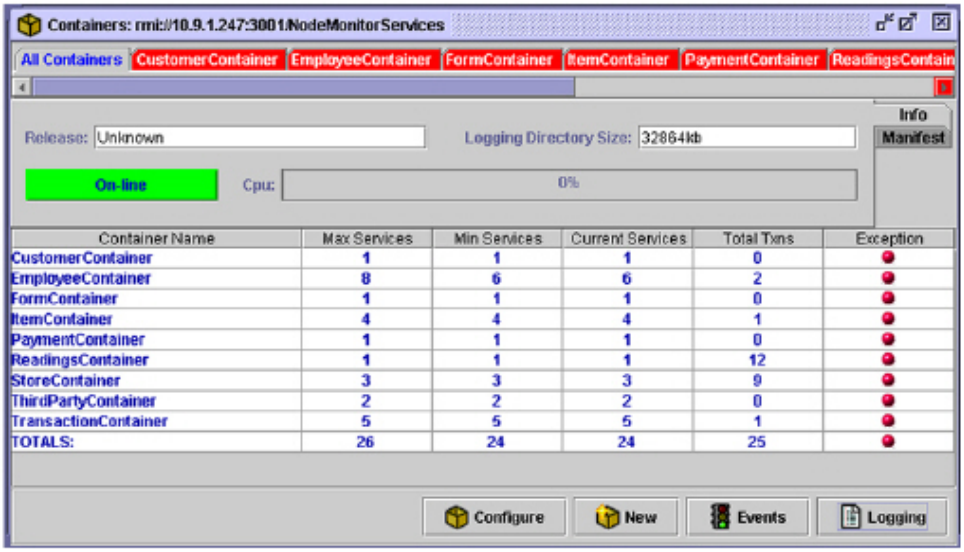
Containers and components	
Containers	**Components**
ThirdPartyContainer	• CreditAuthServices • TaxServices
TransactionContainer	• EODServices • LayawayServices • SOSServices • TransactionPOSServices • TxnPosterServices

Note: Container and service names appear as they would in Mission Control.

Containers window

The **Containers** window provides you with container information, server vitals, and performance information, such as CPU and memory allocation. You can use this information to optimize a server's performance to run the maximum number of concurrent processes without a decrease in performance. Mission Control allows you to view the **Containers** window for all online servers.

When you first connect to a server, the default tab in the **Containers** window is the **All Containers** tab. The **All Containers** tab is unique because it displays information about the entire system and all the containers. The other container-specific tabs display information for the selected container.



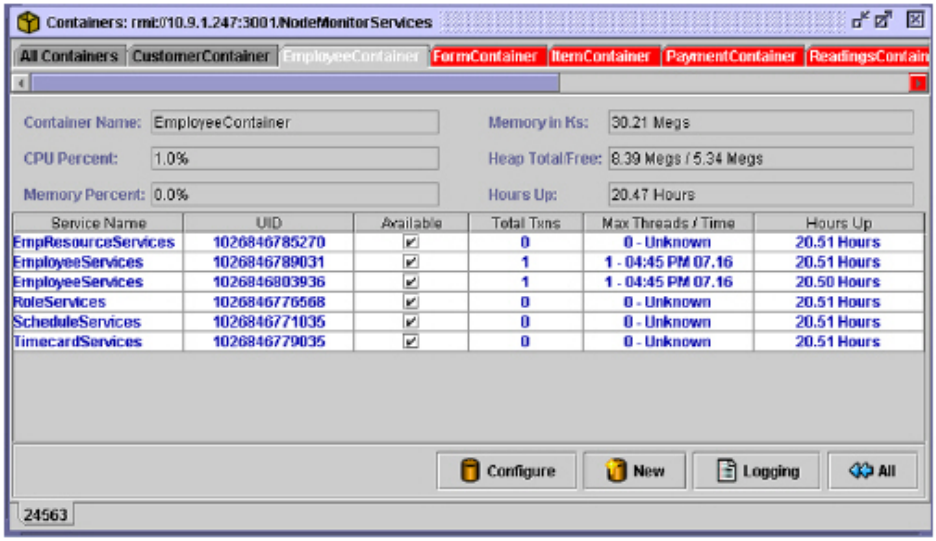
Containers window with the All Containers tab selected

The table below lists and describes the features and information displayed on the **Container** window when the **All Containers** tab is selected.

Container Window Contents (All Containers)	
Feature	**Description**
Tabs	Tabs for containers.
Release	The release number or version of RPOS running on the application server.
Logging Directory	Displays the file size of the log files.

Container Window Contents (All Containers)	
Feature	**Description**
Online/Offline	Displays the online status of the server. The status button provides the following information: • The **Online/Offline** button appears green if the system is online. • The **Online/Offline** button appears red if the system is offline.
CPU	Displays the total CPU usage of a server.
Container table	Lists and displays the following container information: • List of containers • Number of maximum services • Number of minimum services • Number of current services running • Total transactions per container • Status of exceptions—green if there are no exceptions and red if there exceptions logged.
ConfigureContainer button	Allows you to re-configure the selected container.
Events button	Allows you to view event logging for the connected node.
New Container button	Allows you to create and configure a new container
View Container Logging button	Allows you to view the log files of the selected container.

By selecting a container tab in the **Containers** window, you can the view the components within a container. After you select a container tab, the system displays information for the selected container and not the entire server. The image below shows the **Containers** window when a container tab is selected.



Containers window with a container tab selected

The table below lists and describes the features and information displayed on the **Container** window when the a container tab is selected.

Container Window Contents (single container)	
Feature	**Description**
Container Name	Displays the name of the selected container.
Memory in Ks	Displays the actual amount of memory of a container.
CPU Percent	Displays the percentage of CPU used to run a component.
Heap Total/Free	Displays the amount of heap memory used and the amount of available free memory.
Memory Percentage	Displays the percentage of CPU used to run a component.
Hours Up	Displays the number of hours the component has been running.

Container Window Contents (single container)	
Feature	**Description**
Service table	Lists and displays the following container information: • Services—List of components or services within the selected container. • UID—Displays the components unique ID (timestamp of when the component started) of each component. • Availability—Displays the availability status of a component. • Total Txns—Displays the total number of transaction against a component. • Maximum Threads/Time—Displays the maximum number of concurrent threads and the time and date this occurred. • Hours Up—Displays the total number of hours a component has been running.
Configure Component button	Allows you to re-configure the selected component.
New Component button	Allows you to create and configure a new instance of a component.
View Component Logging button	Allows you to view the log files of the selected component.
All Container button	Displays the information for all the containers. Selecting the All Containers button has the same function as selecting the All Container tab.
UID tab	Displays the unique ID (UID) of the container.

Configure a container

The **Configure Container** feature allows you to change a container's settings. Configuring an existing container offers you several ways to monitor a system's performance so you can view and predict performance trends. Viewing a system's statistics and performance allows you to re-configure a component or add a new instance of a component or container so you can balance the workload of components, allocate resources across a system to avoid overloads, resolve exceptions, and debug the system. Mission Control allows you to re-configure the following information:

- Number of instances of a component
- Stats parameter
- Number of default instances
- Time between instance pauses
- Hours of history
- Time between vital pauses

To re-configure the container settings:

1. Connect to a server.
2. In the **Container** table, click the container you wish to configure.

Note: If you do not select a container, the **Select Container** dialog box is displayed. To close the dialog box, click **OK**.

3. Select **Container>Configure**. The **Configure Service Container** window is displayed.
4. Re-configure the container setting.

Add an instance of a component

Adding a new instance of a component is discussed in greater detail in Chapter 4, "Working with Components.".

Change the stats parameter

The stats parameter is the class that collects performance information for each container. The stats parameter should not be changed unless there is an error in the initial settings. For example, this class may change depending on the operating system, and this will require you to make modifications to the configuration file on the application server.

Note: These steps assume you have performed the steps described in "Configuring a Container".

To change the stats parameter:

1. In the **Configure Service Container**, click a stats class. The stats parameter is displayed in the **Argument** field.
2. Highlight and delete the information after **DSTATS_CLASS=**
3. Click the **Class** button. The **Choose Class** window is displayed.
4. In the **Choose Class** window, drill down to `com_iso_cr_node`.
5. Select the new stats parameter. Choose from the following parameter options:
    - WinProcessStats—used for Windows-based operating systems.
    - UnixProcessStats—used for most UNIX- and Linux-based operating systems.
    - UnixSparcProcessStats—used for the UNIX Sparc operating system.
7. In the **Choose Class** window, click **OK**. The **Argument** field is updated with the new stats parameter.
8. In the **Configure Service Container** window, click the **Save** button.
9. To update and save the changes to the stats parameter, click **OK**.

Note: You must click **OK** to update the system with the new stats parameter. Selecting **Save** does not update the system with the new stats parameter.

Modify the default number of instances of a container

Creating a new instance of a container creates a mirror of an existing process. A new instance of a container allows you to manage an increase in workload and helps distribute the workload among multiple instances of a particular container. While increasing the number of instances of containers will increase the CPU used, increasing the number of services can also create a work overload for the container. This work overload bottlenecks the CPU used, and it slows the system. In this case, creating a new instance of a container balances the workload for both the container and components, and it extends the life of the containers and components.

Note: These steps assume you have performed the steps described in "Configuring a Container".

To add a new instance of a container:

1. In the **Configure Service Container** window, type the number of instances of a container in the **Default Instance Count** field.
2. Click **OK**. The new instance of a container are added after the next instance update.

View the new instance of a container

After the you add a new instance of a container and the system updates your changes, Mission Control does not display a new container tab for that instance. Instead, Mission Control generates a new UID for that container and adds the UID to the right of the existing container UID. All container UID's are found at the bottom left of the **Containers** window.

To view the new instance of a container:

1. Select the container tab of the container you created a new instance for.

Note: By default, the **Container** window displays the container information for the container you added a new instance of.

2. Select the UID of the new container. The container information is displayed in the **Containers** window.

Modify the instance pause

Mission Control automatically updates the number of instances of a container after each instance pause. The value in the **Instance Pause in Minutes** field represents the number of minutes in which Mission Control updates the number of instances of each container.

Note: These steps assume you have performed the steps described in "Configuring a Container".

To modify the instance pause value:

1. In the **Configure Service Container** window, type the new number of instances in the **Instance Pause in Minutes** field.
2. Click **OK**. The instance pause is updated after the next instance pause occurs.

Modify the history value

Mission Control allows you to view and track system performance using the **Graphing Vitals** feature. You can modify the **Hours of History** value for graphing purposes only, and this value represents the number of hours of graphing information that is displayed on the **Graphing Container** window.

Note: These steps assume you have performed the steps described in "Configuring a Container".

To modify the hours of history value:

1. In the **Configure Service Container** window, type the new number of instances in the **Hours of History** field.
2. Click **OK**. The system updates the hours of history and displays the changes in the **Graphing Container** window.

Modify the vital pauses

Mission Control automatically updates the vital pauses for graphing purposes. The value in the **Vital Pause in Minutes** field represents the time increments (in minutes) in which the performance statistics are recorded and displayed in the **Graphing Container** window.

Note: These steps assume you have performed the steps described in "Configuring a Container".

To modify the vital pauses: value

1. In the **Configure Service Container**, type the new number of instances in the **Vital Pause in Minutes** field.
2. Click **OK**. The vital pauses is updated, and the changes are displayed in the **Graphing Container** window.

Create a new container

Creating a new container is different than starting a new instance of an existing container. The **Container Wizard** provides step-by-step instructions to create a new container. You will need to provide the following information to create a new container:

- Unique container name
- Default settings
- Specify process stats classname
- Specify stats parameter

To create a new container:

1. Connect to a server.
2. Select **Container>New**.
   OR
   Click the **New Container** button.

   The Containers Wizard is displayed.
3. Follow the step-by-step instructions provided by the **Containers Wizard** to create a new container.

Start and stop an instance of a container

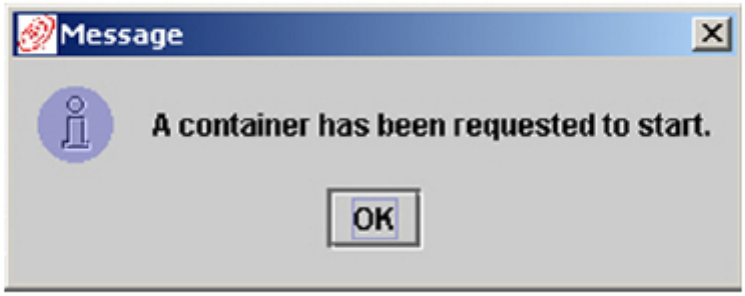
Mission Control allows you to monitor CPU performance and memory allocation so you can document and monitor server performance limitations. Using Mission Control, you can configure the server to spawn a new instance of a container when the threshold for concurrent threads is reached. However, if the number of transactions creates a load too large to be sufficiently handled by the number of concurrent threads, server performance will decrease below the optimal performance levels.

Start a new instance of a container

It is recommended that you start a new instance of a container when a container cannot sufficiently handle the number of concurrent threads without a decrease in server performance. A new instance will help manage the increased workload and number of transactions. Once the workload has decreased to the point where the default number of containers can sufficiently handle the workload, Mission Control automatically stops the oldest instance of the container to help with CPU utilization.

To start a new instance of a component:

1. Connect to a server.
2. In the **Container** table, click the container you wish to start a new instance of.
   OR
   Click the container tab you wish to start a new instance of.
3. Select **Container>Start**. The **Start Service Container** dialog box is displayed.



4. In the **Start Service Container** dialog box, click **OK**. The new instance the selected container is started and a UID is assigned to the new instance of the container.

View the New Instance of a Container

After the you add a new instance of a container and the system updates your changes, Mission Control does not display a new container tab for that instance. Instead, Mission Control generates a new UID for that container and adds it to the left of the existing container UID. Multiple instances of a container can be accessed from a single container tab.

To view the new instance of a container:

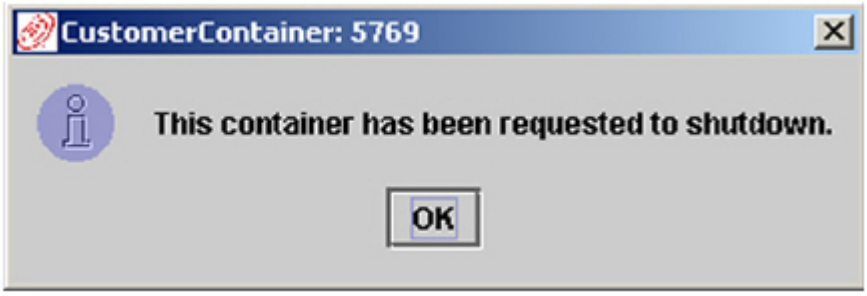
1. Select the container tab for the container you created a new instance for.
2. Select the UID of the new container. The selected container's information is displayed in the **Containers** window.

Stop an instance of a container

After the workload for a component has decreased, you can stop an instance of a container to help allocate and better utilize CPU and system resources. Also, stopping an instance of a container allows developers to incorporate code changes and re-initialize the environment.

To stop an instance of a container:

1. Connect to a server.
2. In the **Container** table, click the container you wish to stop an instance of.
   OR
   Click the container tab of the container you wish to stop an instance of.
3. Select **Container>Shutdown**. The **Shutdown Service Container** dialog box is displayed.



4. In the **Shutdown Service Container** dialog box, click **OK**. The instance the selected container is shut down.

Delete a container

Deleting a container removes the container, components within the container, and the configuration file for the selected container from the server.

To delete a container:

1. Connect to a server.
2. In the **Container** table, click the container you wish to delete.
   OR
   Click the container tab of the container you wish to delete.
3. Select **Container**>**Delete**. The system displays the **Warning** dialog box.
4. In the **Warning** dialog box, click **OK**. The system shutdowns the instance the selected container, and displays the **Shut Down** dialog box.

   Note: Clicking **OK** deletes the configuration file but the processes still reside on the server.

5 In the **Shut Down** dialog box, click **Yes**. The system removes the processes from the server.

   Note: Selecting **No** allows the processes to reside on the server until the server is restarted.

View container log files

Mission Control stores performance statistics and errors in log files. You can view these log files to monitor and manage server performance. For more information on viewing container log files, see Monitor Vitals and View Log Files.

Chapter 4 – Working with components

Components are groupings of functionalities similar to objects in object-oriented programming. The components in Retek ® Point-of-Service™ (RPOS) are implemented by service classes. For example, the customer component is implemented by the customer service. The component, or services, are stored in containers, and the processes are run from the containers. You can have multiple instances of a component within a single container.

You can perform the following tasks when working with components:

- View component properties
- Create a new component
- Configure an existing component
- Start and stop an instance of a component
- Make all components available or unavailable
- Delete a component
- View component log files

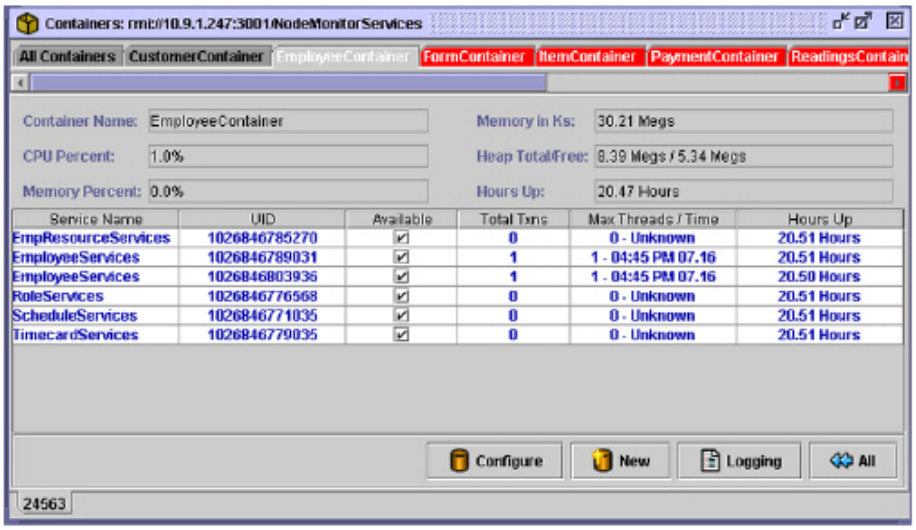
Containers and components are configurable per the customer. The table below lists the default components and the containers that store them within the RPOS application.

Components and Containers	
Components	**Containers**
CreditAuthServices	ThirdPartyContainer
CustomerServices	CustomerContainer
EmpResourceServices	EmployeeContainer
EmployeeServices	EmployeeContainer
EODServices	TransactionContainer
FormServices	FormContainer
GoalingServices	StoreContainer
InventoryServices	ItemContainer
ItemServices	ItemContainer
LayawayServices	TransactionContainer

Components and Containers	
Components	**Containers**
MerchandiseServices	ItemContainer
ReadingServices	ReadingsContainer
RedeemServices	PaymentContainer
RegisterServices	ReadingsContainer
RoleServices	EmployeeContainer
ScheduleServices	EmployeeContainer
StoreServices	StoreContainer
TaxServices	ThirdPartyContainer
TimecardServices	EmployeeContainer
TransactionPOSServices	TransactionContainer
TransferServices	ItemContainer
TxnPosterServices	TransactionContainer

View the container window

The **Container** window not only provides you with container information, it also provides information about the components or services running on the servers. When you first connect to a server, the default tab in the **Containers** window is the **All Containers** tab. The **All Containers** tab is unique because it displays information about the entire system and all the containers where the other tabs display information for the selected container. By selecting a container tab in the **Containers** window, you can the view the components within a container.



Container window with a container tab selected

The table below lists and describes the features and information displayed on the **Container** window when the a container tab is selected.

Container Window Contents (single container)	
Feature	**Description**
Container Name	Displays the name of the selected container.
Memory in Ks	Displays the actual amount of memory used by a container.
CPU Percent	Displays the percentage of CPU used to run a component.
Heap Total/Free	Displays the amount of heap memory used and the amount of available free memory.
Memory Percentage	Displays the percentage of CPU used to run a component.
Hours Up	Displays the number of hours the component has been running.

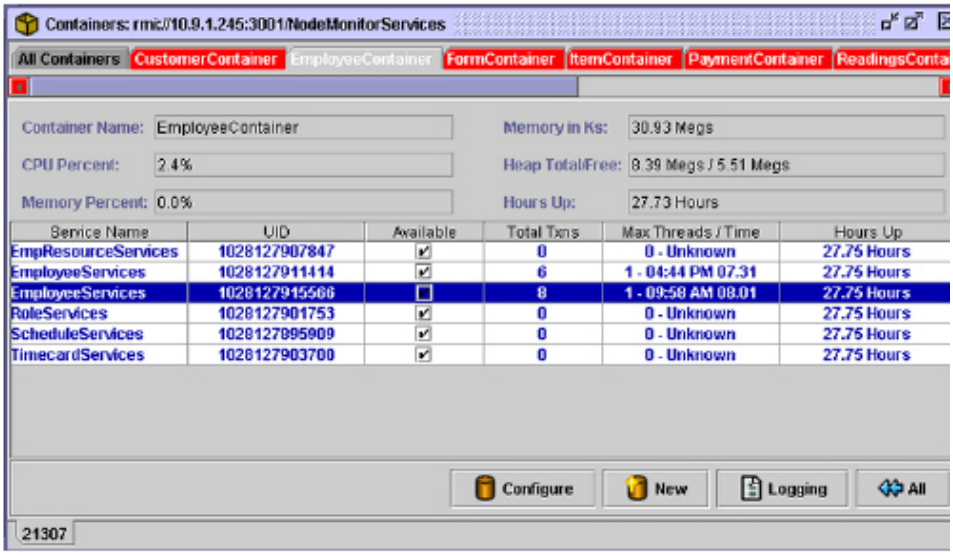
Container Window Contents (single container)	
Feature	**Description**
Service table	Lists and displays the following container information: • Services—List of components or services within the selected container. • UID—Displays the components unique ID (timestamp of when the component started) of each component. • Availability—Displays the availability status of a component. • Total Txns—Displays the total number of transaction against a component. • Maximum Threads/Time—Displays the maximum number of concurrent threads and the time and date this occurred. • Hours Up—Displays the total number of hours a component has been running.
Configure Component button	Allows you to re-configure the selected component.
New Component button	Allows you to create and configure a new instance of a component.
View Component Logging button	Allows you to view the log files of the selected component.
All Container button	Displays the information for all the containers. Selecting the All Containers button has the same function as selecting the All Container tab.
UID tab	Displays the UID of the container. If there are multiple instances of a container, there will be a UID tab for each instance.

View component properties

After connecting to a server, you can view and manage a servers performance by monitoring the components running on that server.

To view components running on a server:

1. Connect to a server. The **Containers** window is displayed.
2. In the **Containers** window, click the container tab you wish to create the new component in.
3. Select a component. The selected component is highlighted in blue.



Note: If you do not select a component, the **Select Component** dialog box is displayed.

4. Select **Component>Configure**. The **Configure Component** dialog box is displayed.
5. In the **Configure Component** dialog box, view the component properties. The following is a list of component properties displayed in the **Configure Component** dialog box:
   - Component name
   - Class name
   - Key-Value pairs
   - Minimum number of instances
   - Maximum number of instances

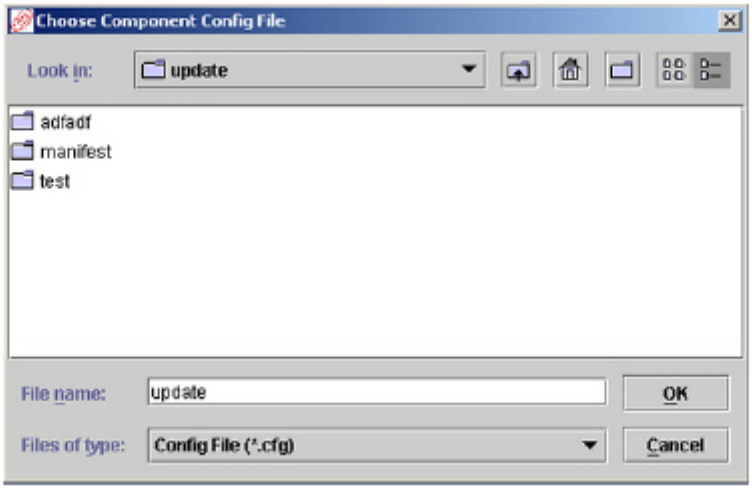
Create a new component

Mission Control allows you to create a new component if the service does not already exist. Before you can create a new component using Mission Control, you must generate a service using RPOS Tools Application Builder. For more information on generating a new service, see the *RPOS Tools Application Builder User Guide*.

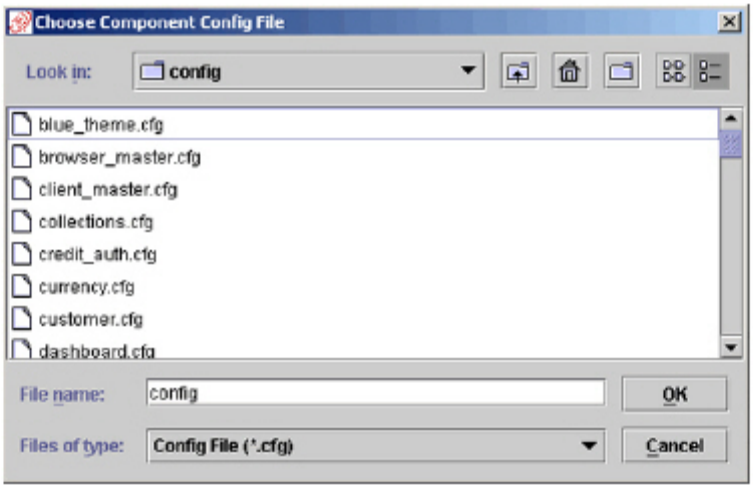
To create a new component:

1. Connect to a server. The **Containers** window is displayed.
2. In the **Containers** window, click the container tab that you will create the new component in.
3. Select **Component>New**.
   OR
   Click the **New Component** button.

The **Choose Component Config File** dialog box is displayed.



4. Drill down to `iso_store>file>prod>config`.



5. Select the configuration file you created using Application Builder.

📖 **Note:** Configuration files have a **.cfg** extension.

6. Click **OK**. The **Add Component** dialog box is displayed.

📖 **Note:** When you select a configuration file, a default value is given to the component name and classname.

📖 **Note:** The default value for **Minimum Instance** and **Maximum Instance** is 1.

7. If necessary, configure the component.
8. In the **Add Component** dialog box, click **OK**.

Configure a component

The **Configure Component** feature allows you to change the properties of a component. Mission Control allows you to re-configure the following component properties:

- Configuration file
- Class name
- Number of minimum instances
- Number of maximum instances

Modify the configuration file

You should not have to re-configure the configuration file of a component unless you are pointing to the wrong configuration. If any modifications need to made, they are normally made within the configuration file itself using Application Builder.

To modify the configuration file of a component:

1. Connect to a server. The **Containers** window is displayed.
2. In the **Containers** window, click the container tab you wish to modify.
3. Select **Component>Configure**. The **Configure Component** dialog box is displayed.
4. In the **Component Properties** area, click the configuration file name. The configuration file is displayed in the **Key-Value Pairs** field.
5. In the **Key-Value Pairs** field, highlight and delete the information after **CONFIG_FILE=**.
6. Click the **Configuration File** button. The **Choose Configuration File** window is displayed.
7. In the **Choose Configuration File** window, click the new configuration file.
8. Click **OK**. The **Key Value Pairs** field is updated.
9. In the **Configure Component** dialog box, click the **Save** button.
10. To save the changes to the component and update the system, click **OK**.

Note: You must click **OK** to update the component with the new configuration file. Clicking the **Save** button does not update the component with the new configuration file.

Modify the class name

Modifying the class name modifies the service that the component runs. When modifying the class name, you must select a class that extends the CMS Component.

To modify the class name of a component:

1. Connect to a server. The **Containers** window is displayed.
2. In the **Containers** window, click the container tab you wish to modify.
3. Select the component you wish to change the class name of.
4. Select **Component>Configure**. The **Configure Component** dialog box is displayed.

Note: If have not selected a component, the **Select Component** dialog box is displayed. To close the dialog box, click **OK**, and select a component before you begin modifying the classname.

5. In the **Component Properties** group box, click the **Specify Class Name** button. The **Select Class** window is displayed.
6. In the **Select Class** window, drill down to `com_iso_cr` and select the appropriate class name.
7. Click **OK**. The system updates the **Class Name** field with the selected class name.
8. To update the component with the new configuration file, click the **Save** button.
9. To save the changes to the component and update the system, click **OK**.

Note: You must click **OK** to update the component with the new configuration file. Selecting **Save** does not update the component with the new configuration file.

Modify the number of component instances

Mission Control allows you to modify the number of component instances so you can better manage the workload of each component and container.

Modify the minimum number of component instances

To modify the minimum number of component instances:

1. Connect to a server. The **Containers** window is displayed
2. In the **Containers** window, click the container tab you wish to modify.
3. Select **Component>Configure**. The **Configure Component** dialog box is displayed.
4. In the **Component Instance** area, type the number of minimum instances in the **Minimum Instances** field.
5. Click **OK**. If current number of instances for this component is less than the minimum number of component instances, new instances of this component will be added after the next instance pause.

Modify the maximum number of component instances

Modifying the maximum number of component instances allows you to start new instances of a component when the workload is heavy.

To modify the maximum number of component instances:

1. Connect to a server. The **Containers** window is displayed.
2. In the **Containers** window, click the container tab you wish to modify.
3. Select **Component>Configure**. The **Configure Component** dialog box is displayed.
4. In the **Component Instance** area, type the number of minimum instances in the **Maximum Instances** field.
5. Click **OK**.

Start and stop an instance of a component

Mission Control allows you to monitor CPU performance and memory allocation so server performance limitations can be documented. Using Mission Control, you can configure the server to spawn a new instance of a component when the threshold for concurrent threads is reached. However, if the number of transactions creates a workload too large to be sufficiently handled by the number of concurrent threads, server performance will decrease below the optimal performance levels.

Start a new instance of a component

It is recommended that you start a new instance of a component when the number of concurrent threads cannot sufficiently handle the current workload. A new instance will help manage the increased workload and number of transactions.

To start a new instance of a component:

1. Connect to a server. The system displays the **Containers** window.
2. In the **Containers** window, click the container tab that stores the component you wish to start a new instance of.
3. Select the component you wish to start a new instance of. The selected component is highlighted in blue.
4. Select **Component>Start**. The **Start Instance** dialog box is displayed.
5. In the **Start Instance** dialog box, click **OK**. The system starts a new instance of the selected component after the next instance pause and assigns it a UID after the next instance pause.

Stop an instance of a component

After the workload for a component has decreased, you can stop an instance of that component to help allocate memory back to the system.

To stop an instance of a component:

1. Connect to a server. The **Containers** window is displayed.
2. In the **Containers** window, click the container tab that stores the component you wish to stop an instance of.
3. Select the component you wish to stop an instance of. The selected component is highlighted in blue.
4. Select **Component>Shutdown**. The **Shutdown Instance** dialog box is displayed.
5. In the **Shutdown Instance** dialog box, click **OK**. The instance of the selected component is stopped after the next instance pause.

Using the availability feature

The **Available** option allows you to make all services available or not available. The Available option can be found on the **File** menu or on the **Containers** window. During database or server maintenance, services must be taken offline. This option allows you to make all services not available without having to turn off each service one at a time.

Note: After connecting to a server, all services are available to the users and the **Available** option is selected.

Making all services not available

To make all services not available:

Note: The default mode in Mission Control is to have all the services running on a server.

1. Connect to a server. The **Containers** window is displayed.
2. Select **File>Node>Available**.
   OR
   Click the **Available** button on the **Containers** window. The **All Services Not Available** dialog box is displayed.
3. In the **All Services Available** dialog box, select **OK**. All the service running on the server are now unavailable.

Making all services available

To make all services available:

1. Connect to a server. The **Containers** window is displayed.
2. Select **File>Node>Available**.
   OR
   Click the **Available** button on the **Containers** window. The **All Services Available** dialog box is displayed.
3. In the **All Services Available** dialog box, select **OK**. The **Container** window is updated and all services become available.

Deleting an existing component

To delete an exiting component:

1. Connect to a server. The **Containers** window is displayed.
2. In the **Containers** window, click the container tab that stores the component you wish to delete.
3. Select the component you wish to delete.
4. Select **Component Delete**. The **Warning** dialog box is displayed.
5. In the **Warning** dialog box, click **OK**.
6. In the **Components** table, verify that Mission Control has deleted the component.

Viewing component log files

Mission Control stores performance statistics and errors in log files. You can view these log files to monitor and manage server performance. For more information on viewing component log files, see Chapter 7, “Monitor Vitals and View Log Files”.

Chapter 5 – Working with clients

Retek ® RPOS™ Tools Mission Control allows you to monitor client running the RPOS system from a remote location. After connecting to a client, you can perform the following tasks using Mission Control:

- View client statistics
- Get broken transactions
- Launch RPOS Tools Object Inspector application tool to fix broken transactions
- Remove broken transactions
- Clear broken transactions
- Clear all log exceptions
- View client peers
- Shut down client

Client browser window

The **Client Browser** window provides you with three tabs that allows you to monitor a client:

- **Current Statistics** tab—allows you to view performance statistics and properties of the client.
- **Broken Transactions** tab—allows you to view and retrieve broken transactions from the client.
- **Known Peers** tab—allows you to view the other clients within the store or connected to the local area network (LAN).

Note: The default tab when opening a **Client Browser** window is the **Current Statistics** tab.

Note: The **Current Statistics** tab is the default tab when opening a **Client Browser** window.

The table below lists and describes the features and information displayed on the **Client Browser** window.

Client Browser Window Contents		
Feature	**Tab**	**Description**
IP Address	Current Statistics	Displays the IP address of the selected client terminal.
Process Type	Current Statistics	Displays the node type. A process type with a value of **2** designates a client terminal.
Store ID	Current Statistics	Displays the store ID number (unique for each store).

Client Browser Window Contents		
Feature	**Tab**	**Description**
Register ID	Current Statistics	Displays the register ID number of the selected client terminal (unique for each store).
Pending TXN	Current Statistics	Displays the number of pending transactions for the selected client terminal.
Broken TXN	Current Statistics	Displays the number of broken transactions for the selected client terminal.
Shutdown Client button	Current Statistics	Allows you to shutdown processes on the selected client terminal.
View Client Logging button	Current Statistics	Allows you to view the log files of the selected client terminal.
Broken Transactions display area	Broken Transactions	Displays the broken transactions on the selected client terminal.
Directory	Broken Transactions	Displays the file directory for broken transactions.
Get Txn button	Broken Transactions	Allows you to retrieve broken transactions.
Remove Txn button	Broken Transactions	Allows you to remove broken transactions on the selected client terminal.
Clear Txn button	Broken Transactions	Allows you to clear broken transactions on the selected client terminal.
Known Peers display area	Known Peers	Allows you to view peers of the selected client terminal.

Broken transactions

Mission Control allows you to work with broken transactions that reside on the clients. To fix a broken transaction, you will have to open Object Inspector. Mission Control allows you to perform the following tasks regarding broken transactions:

- Retrieve a broken transaction
- Save a broken transaction
- Clear all broken transactions

Retrieve a broken transaction

Mission Control allows you to retrieve the details of the broken transaction from the client and either open Object Inspector to fix the transaction or save the transaction on your hard drive so you can fix it at a later time.

To get a broken transaction:

1. Connect to a client. The **Client Browser** window is displayed.
2. Click the **Broken Transaction** tab.
3. In the **Broken Transaction** display area, select a broken transaction. The selected transaction in blue is highlighted.
4. Click the **Get Txn** button.
   OR
   Select **Client>Get Transaction**. The **Get Transaction** display box is displayed.

Note: If you do not select a broken transaction, the system displays the **Please Select Transaction** dialog box. To close the dialog box, click **OK**.

5. To retrieve the broken transaction and launch Object Inspector, click **Yes**.
   OR
   To save the broken transaction onto your hard drive, click **No**.

Remove a broken transaction

After you retrieve details about a broken transaction, you can remove the transaction from the client terminal.

To remove a broken transaction:

1. Connect to a client. The **Client Browser** window is displayed.
2. Click on the **Broken Transaction** tab.
3. In the **Broken Transaction** display area, select a broken transaction. The selected transaction is highlighted in blue.
4. Click the **Remove Txn** button.
   OR
   Select **Client>Remove Transaction**. The **Remove Broken Transaction** display box is displayed.
5. In the **Remove Broken Transaction** display box, click **Yes**. The transaction is removed from the client terminal, and it is no longer displayed in the **Client Browser** window.

Clear all broken transactions

After you retrieve details about a broken transaction, Mission Control allows you to remove all broken transactions from the client terminal at one time. The **Clear Txn** button allows you to clear all client errors from client terminal, and it deletes all files in the `client.err` folder.

To remove all broken transactions:

1. Connect to a client. The **Client Browser** window is displayed.
2. Click on the **Broken Transaction** tab.
3. In the **Broken Transaction** display area, select a broken transaction. The selected transaction is highlighted in blue.
4. Click the **Clear Txn** button.
   OR
   Select **Client>Get Transactions**. The **Clear Transactions** display box is displayed.
5. In the **Clear Transactions** display box, click **Yes**. The transaction is removed from the client terminal, and the transaction is longer displayed in the **Client Browser** window.

View client peers

The **Client Browser** window allows you to view other client terminals within the store or LAN. However, you cannot connect to them from this window. You will have to connect to a new client terminal using the **Monitor Node** feature. In the **Known Peers** tab, a display area shows the IP addresses of the client peers so you can use that information to connect to another client terminal within the store or LAN.

To view client peers:

1. Connect to a client. The **Client Browser** window is displayed.
2. Click the **Known Peers** tab. The **Known Peers** display area is displayed with the IP addresses of client terminals on the same LAN.

Clear log exceptions

Clearing log exceptions clears all client errors and deletes items in the `client.err` folder.

To clear log exceptions:

1. Connect to a client. The **Client Browser** window is displayed.
2. Select **Client>Clear Exceptions**.

Shut down a client

Mission Control allows you to shut down a client terminal from a remote location.

To shut down a client:

1. Connect to a client. The **Client Browser** window is displayed.
2. Click the **Shutdown** button.
   OR
   Select **Client>Shutdown**. The **Shutdown Client** display box is displayed.
3. In the **Shutdown Client** display box, click **Yes**. The client is shutdown and the client is no longer displayed in the **Known Peers** display area.

View client log files

Mission Control stores performance statistics and errors in log files. You can view these log files to monitor and manage server performance. For more information on viewing container log files, see Chapter 7, “Monitor Vitals and View Log Files”.

Chapter 6 – Working with the naming server

Retek ® RPOS™ Tools Mission Control allows you to view and monitor the remote naming server (RNS). The RNS serves as the middleware in the RPOS Platform, and it is responsible for storing references to remote objects. The clients will call on the RNS for references to services that are running on the application server. The naming service will then distribute these services to the clients. Basically, the RNS is a communication bridge between the clients and the application servers.

Each service corresponds to a remote reference that is stored on the RNS. When the RNS distributes a remote reference, it is really distributing a reference to a component within a container. These components are the RPOS services that the application servers provide to the clients.

Note: The RNS can be a standalone server or reside on a master naming server or application server.

Overview of the naming browser window

The **Naming Browser** window provides you with a list of components and the properties of the selected naming server. The **Naming Browser** window consists of the following tabs:

- **Components** tab—displays information about the available components.
- **Properties** tab—displays properties of the naming server.

Note: The **Components** tab is the default tab.

Components tab

The **Components** tab displays information about the available components running on the application servers. The **Components** tab provides the following information:

- **Component name**—displays the name of all the components registered with the RNS.
- **Version**—displays the version of the components. The version number of a component is updated with each manifest or update.
- **Instances**—displays the available number of instances of each component.

Properties tab

The **Properties** tab displays general information of the connected naming server. The **Properties** tab provides the following information:

- **Debug**—displays whether or not the additional logging output feature is selected or not.
- **Redirect Output**—displays whether or not server activity will be sent to the log files or the console.
- **Monitor**—displays that multi-thread monitoring of components registered with the RNS on. This option will spawn threads for each component and issue a ping. Also, the RNS will distribute the workload balance and requests by randomly selecting available services on the application server.
- **RNS Port**—displays the port listening for TCP.
- **Vital Pause**—displays the time between instance pauses.
- **Hours History**—displays the amount of time that log files are recorded.
- **Socket Timeout**—displays the amount of time before a socket times out.

Chapter 7 – Monitor vitals and view log files

Retek ® RPOS™ Tools Mission Control gathers and saves performance statistics for all server processes. Mission Control also stores exceptions and errors in log files so you can view these log files in order to monitor and manage the performance of all the nodes on the network. Mission Controls has the following features that allow you to view performance statistics, exception errors, and log files:

- Graphing component and container vitals
- Percentage chart of components and containers
- Log file with four levels of messages and exception errors

Graph system vitals

The **Graph Component Vitals** feature allows you to monitor, manage, and tune your system for the optimal performance levels. Mission Control allows you to view graphs for the following three levels:

- Container vitals—displays container vitals in the **Graphing Container** window.
- Component vitals—displays component vitals in the **Graphing Component** window.
- Component performance vitals—displays performance vitals for the selected component in the **Graphing Performance** window.

Each graphing level has its own unique window that opens within that displays system vitals specific to that level.

Graphing windows overview

There are three graphing windows that correspond to each graph level. The following is a list of graphing windows:

- **Graphing Container** window—displays graphing information for container vitals.
- **Graphing Component** window—displays graphing information for component vitals.
- **Graphing Performance** window—displays graphing information for performance vitals.

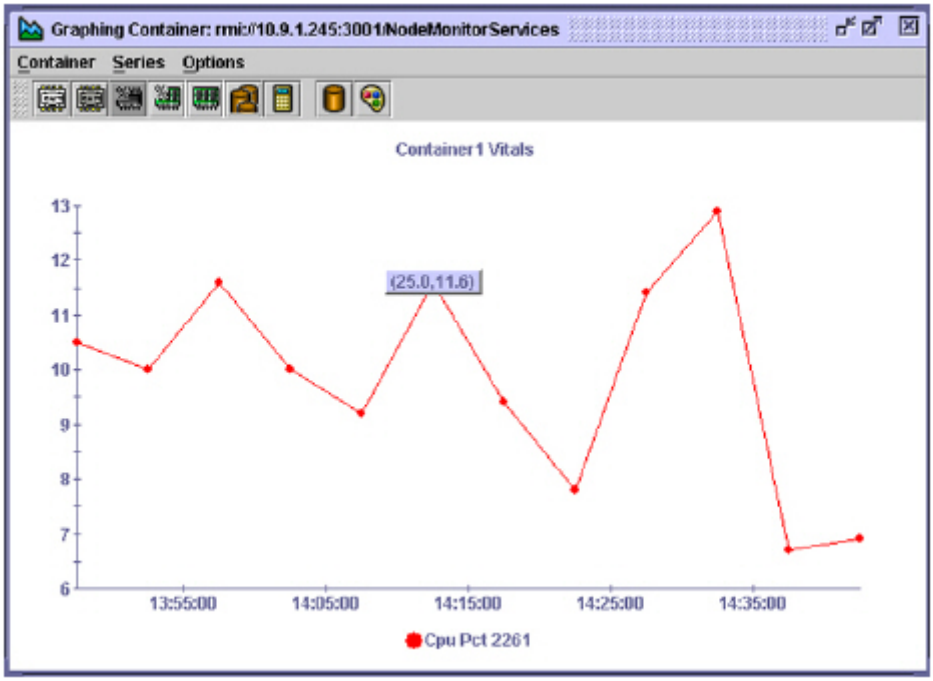
Graphing container window

The **Graphing Container** window displays graphing information for container vitals. You can connect and view the **Graphing Container** window for multiple containers. Each time you wish to view the vitals for a container, Mission Control displays a new **Graphing Container** window.

The **Graphing Container** workspace is made up a menu bar, toolbar, and the vitals graph. The following menu options are available from the **Graphing Container** window:

- **Container** menu—allows you to connect to open the **Graphing Component** window for a specific component, and it displays the unique ID of the viewable container.
- **Series** menu—allows you to select the container vitals you wish to view.

- **Options** menu—allows you to hide the toolbar and to display the **JClass Chart Properties** window to modify the properties of the graph.



Graphing Container window

Mission Control allows you to view the following container vitals using the graphing feature:

- Free heap memory
- Total heap memory
- CPU percentage
- Memory percentage
- Actual memory
- Component count
- Total transaction count

You can view multiple container vitals at one time. Each time you view a container vital, the vitals graph is updated with a value on the graph. Container values are color coded on the vitals graph. A legend on the bottom of the **Graphing Containers** window helps you distinguish the color coding for each container vital.

The toolbar contains command buttons that display container vitals. The command buttons options for viewing container vitals correspond to the options on the **Series** menu. The table below lists and describes the container vitals and corresponding command buttons on the **Graphing Containers** window.

Container Vitals Options	
Graphing Option Name	**Description**
Free Heap Memory 	To view the free heap memory of a container: • Click the Free Heap Memory button. • Select Series>Free Heap Memory.
Total Heap Memory 	To view the total heap memory of a container: • Click the Total Heap Memory button. • Select Series>Total Heap Memory.
CPU Percentage 	To view the CPU percentage of a container: • Click the CPU Percentage button. • Select Series>CPU Percentage.
Memory Percentage 	To view the memory percentage of a container: • Click the Memory Percentage button. • Select Series>Memory Percentage.
Actual Percentage 	To view the actual percentage of memory of a container: • Click the Actual Percentage button. • Select Series>Actual Percentage.
Component Count 	To view the component count of a container: • Click the Component Count button. • Select Series>Component Count.
Total Transaction Count 	To view the total transaction count of a container: • Click the Total Txn Count button. • Select Series>Total Txn Count.

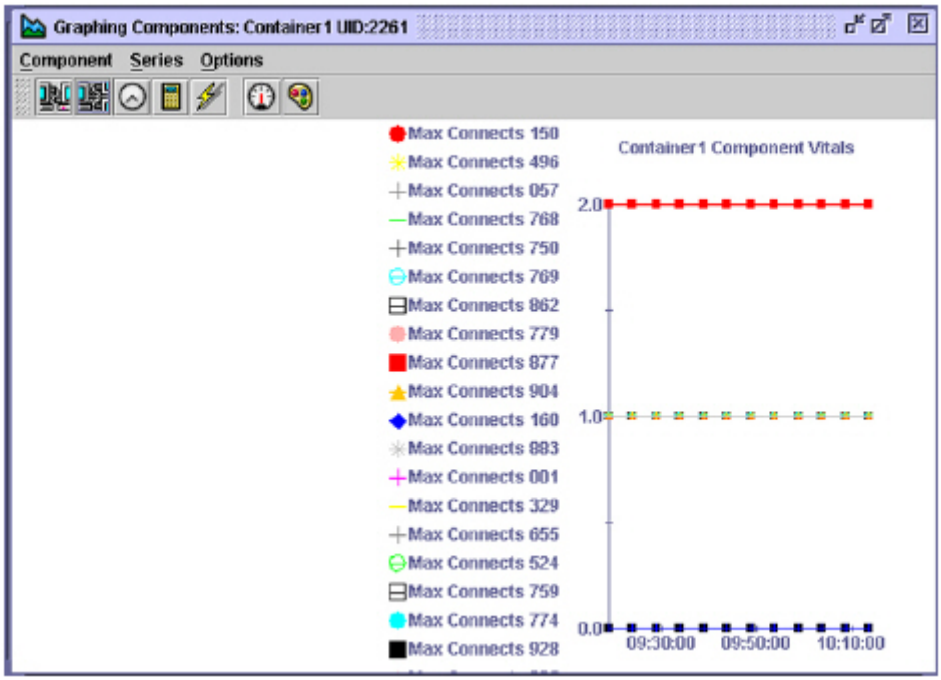
Note: A selected viewing option appears darkened.

Graphing components window

The **Graphing Component** window displays graphing information for component vitals. You can view component vitals for multiple components simultaneously. From the **Graphing Component** window, you can display performance vitals for the components running on the application server.

The **Graphing Component** workspace is made up a menu bar, toolbar, and the vitals graph. The following menu options are available from the **Graphing Container** window:

- **Component** menu—lists all the components running on an application server, and it allows you to select which components you wish to view vitals for. Also, this menu allows you to open the **Graphing Performance** window for a specific component.
- **Series** menu—allows you to select the component vitals you wish to view.
- **Options** menu—allows you to hide the toolbar and to display the **JClass Chart Properties** window to modify the properties of the graph.



Graphing Component window

Mission Control allows you to view the following component vitals using the graphing feature:

- Current connections
- Maximum connections
- Maximum connect time
- Total transaction count
- Transactions per period

You can view multiple component vitals at one time. Each time you view a component vital, the vitals graph is updated with a value on the graph. Component values are color coded on the vitals graph. A legend on the bottom of the **Graphing Components** window helps you distinguish the color coding for each component vital.

The toolbar contains command buttons that display component vitals. The command buttons options for viewing component vitals correspond to the options on the **Series** menu. The table below lists and describes the component vitals and corresponding command buttons on the **Graphing Components** window.

Component Vitals options	
Graphing Option Name	**Description**
Current Connections 	Displays the current connection statistics for the selected components. To display the current connections of a component: • Click the Current Connections button. • Select Series>Current Connections.
Maximum Connections 	Displays the maximum connections statistics for the selected components. To display the maximum connections of a component: • Click the Maximum Connections button. • Select Series>Maximum Connections.
Maximum Connect Time 	Displays the maximum connect time for the selected components. To display the maximum connect time of a component: • Click the Maximum Connect Time button. • Select Series>Maximum Connect Time.
Total Txns Count 	Displays the total transaction count for the selected components. To display the total transaction count for a component: • Click the Total Txns Count button. • Select Series>Total Txns Count.
Txns Per Period 	Displays the transactions per period for the selected components. To display the transactions per period on a graph: • Click the Txns Per Period button. • Select Series>Txns Per Period.

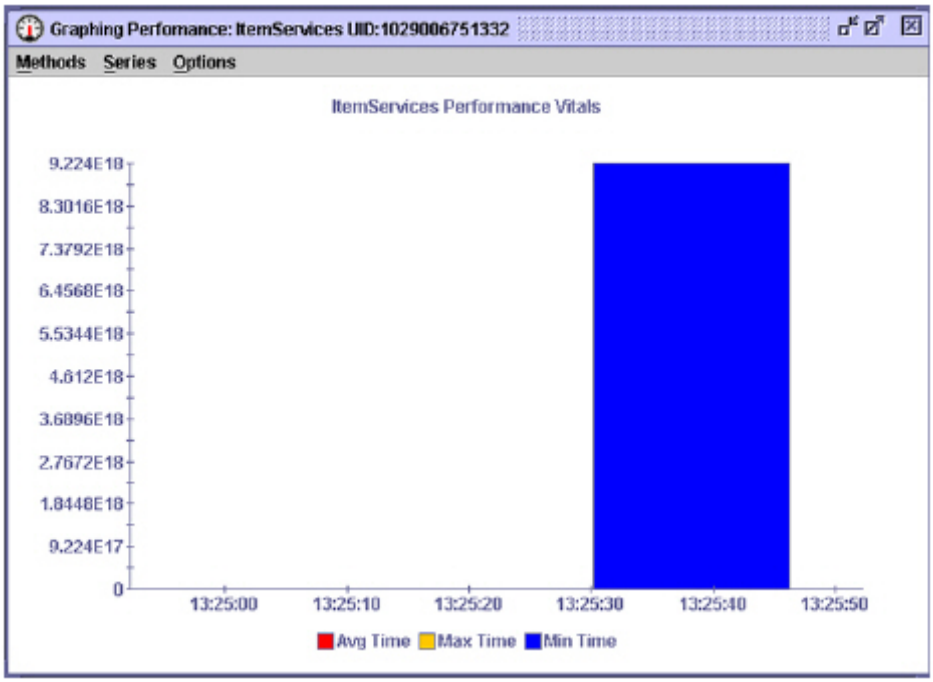
Note: A selected viewing option appears darkened.

Graphing performance window

The **Graphing Performance** window displays graphing information for performance vitals. Performance vitals record how long methods took to execute. The **Graphing Performance** window displays performance vitals for a single component. However if the component is a service with multiple methods within it, you can view the performance vitals of multiple methods simultaneously.

The **Graphing Performance** workspace is made up a menu bar and the performance graph. The following menu options are available from the **Graphing Performance** window:

- **Methods** menu—allows you to connect to the methods within the service.
- **Series** menu—allows you to select the performance vitals you wish to view.
- **Options** menu—allows you to hide the toolbar and to display the **JClass Chart Properties** window to modify the properties of the graph.



Graphing Performance window

Mission Control allows you to view the following component vitals using the graphing feature:

- Average time—records the average time for a method to execute.
- Maximum time—records the maximum time for a method to execute.
- Minimum time—records the minimum time for a method to execute.

Note: Unlike the **Graphing Container** window and **Graphing Component** window, the **Graphing Performance** window does not have a toolbar with command buttons that correspond to the **Series** menu. To view performance vitals for a component, select the **Series** menu and then the performance option.

Graph container vitals

To graph container vitals:

1. Connect to an application server. The **Containers** window for that application server is displayed.
2. On the **Containers** window, select the container you wish to view the vitals for. The selected container is highlighted in blue.
3. Click the **Component Vitals Graph** button.
   OR
   Select **Graph>Vitals**. The **Graphing Container** window is displayed.

Note: If you do not select a container before selecting the graphing feature, the **Select Container** dialog box is displayed. To close the dialog box, select **OK**.

4. From the toolbar, select the appropriate command button for the vital you wish to view.
   OR
   Select **Series** and the appropriate menu option for the vital you wish to view. A line graph representing the vital is displayed on the vitals graph.

Note: For a list of component vitals you can view and monitor, see “Container Vitals Options”.

View multiple container vitals

You can view multiple container vitals simultaneously. Container vitals are color coded so you can easily distinguish between container vitals. Each time you view a container vital, the graph legend displays the selected container vitals and the assigned color.

To view multiple container vitals simultaneously:

1. Connect to an application server and view the **Graphing Container** window.
2. From the toolbar, select the appropriate command button for the vital you wish to view.
   OR
   Select **Series** and the appropriate menu option for the vital you wish to view. A line graph representing the vital is displayed on the vitals graph.
3. To display multiple container vitals simultaneously, repeat step 2. A new line graph representing the selected vital is displayed on the vitals graph along with previously selected vitals.

Deselect container vitals

After you have displayed multiple container vitals for a container, you can deselect a container vital so it no longer is displayed on the **Graphing Container** window. To deselect a container vital:

Note: If a container vital has been selected and is currently displayed on the vitals graph, the command button is darkened and a check is displayed by the menu option on the **Series** menu.

1. Click on the appropriate previously selected command button.
   OR
   Select **Series** and the appropriate previously selected menu option to deselect a container vital. The container vital is no longer displayed on the vitals graph.
2. To deselect more container vitals so they no longer appear on the vitals graph, repeat Step 1.

Show and hide graphing container window toolbar

Mission Control allows you to show or hide the toolbar on the **Graphing Container** window. Remember, the **Series** menu options correspond with the command buttons that appear on the toolbar. When you first connect to the **Graphing Container** window, the toolbar is displayed.

To hide the **Graphing Container** window toolbar:

- Select **Options>Show Toolbar**. The checkbox is deselected , and the toolbar and command buttons are no longer displayed on the **Graphing Container** window.

Modifying chart options

The **Chart Options** menu option allows you to customize the appearance of the vitals graph. The **Chart Options** button starts Sitraka's JClass Chart Customizer that allows you to customize and view the graph properties.

To customize the graph properties:

1. Select **Options>Chart Options**.
   OR
   Click the **Chart Options** button. The **JClass Chart Properties** dialog box is displayed.
2. On the **JClass Chart Properties** dialog box, make the necessary changes to the appearance of the container vitals graph.

Note: Visit the Sitraka website at **//www.sitraka.com** for documentation on the JClass Chart Customizer.

Graph component vitals

To graph component vitals:

1. Connect to an application server. The **Containers** window for that application server is displayed.
2. On the **Containers** window, select the container you wish to view the vitals for. The selected container is highlighted in blue.
3. Select **Graph>Vitals**.
   OR
   Click the **Component Vitals Graph** button. The **Graphing Container** window is displayed.

   Note: If you do not select a container before selecting the graphing feature, the **Select Container** dialog box is displayed. To close the dialog box, select **OK**.
4. Click the **Graph Components** button.
   OR
   Select Container>Graph Components. The Graphing Component window is displayed.
5. From the toolbar, select the appropriate command button for the component vital you wish to view.
   OR
   Select **Series** and the appropriate menu option for the component vital you wish to view. A line graph representing the vital is displayed on the vitals graph.

   Note: For a list of component vitals you can view and monitor, see Table 7-2, “Component Vitals Options".

   Note: By default, the vitals for all components within the selected container are displayed.

Deselect a component

By default, the vitals for all components within a container are selected when you open the **Graphing Components** window. If you graph a component vital, the results for all components within the selected container will be displayed. You can modify the results to display the vitals for only the desired components within a container by deselecting a component so it is no longer displayed on the vitals graph.

To deselect a component:

1. On the **Graphing Components** window, select **Component>Show Component**. A submenu with all the available components is displayed. All listed components are selected with a check mark.
2. On the submenu, select the component you no longer wish to view the component vitals for. Component vitals for the deselected component are no longer displayed on the vitals graph.
3. To deselect multiple components, repeat steps 1-2.

View multiple component vitals

You can view multiple component vitals simultaneously. Component vitals are color coded so you can easily distinguish between component vitals. Each time you view a component vital, the graph legend displays the selected component vitals and the assigned color.

To view multiple component vitals simultaneously:

1. Connect to an application server and view the **Graphing Component** window.
2. From the toolbar, select the appropriate command button for the vital you wish to view.
   OR
   Select **Series** and the appropriate menu option for the vital you wish to view. A line graph representing the vital is displayed on the vitals graph.
3. To display multiple container vitals simultaneously, repeat step 2. A new line graph representing the selected vital is displayed on the vitals graph along with previously selected vitals.

Deselect component vitals

After you have displayed multiple vitals for a component, you can deselect a component vital so it is no longer displayed on the **Graphing Component** window.

To deselect a component vital:

Note: If a component vital has been selected and is currently displayed on the vitals graph, the command button is darkened and a check is displayed by the menu option on the **Series** menu.

1. Click on the appropriate previously selected command button.
   OR
   Select **Series** and the appropriate previously selected menu option to deselect a container vital. The component vital is no longer displayed on the vitals graph.
2. To deselect more component vitals so they are no longer displayed on the vitals graph, repeat Step 1.

Show and hide graphing component window toolbar

Mission Control allows you to show or hide the toolbar on the **Graphing Component** window. Remember, the **Series** menu options correspond with the command buttons that appear on the toolbar. When you first connect to the **Graphing Container** window, the toolbar is displayed.

To hide the **Graphing Component** window toolbar:

- Select **Options>Show Toolbar**. The checkbox is deselected , and the toolbar and command buttons are no longer displayed on the **Graphing Component** window.

Modify chart options

The **Chart Options** menu option allows you to customize the appearance of the vitals graph. The **Chart Options** button starts Sitraka's JClass Chart Customizer that allows you to customize and view the graph properties.

To customize the graph properties:

1. Select **Options>Chart Options**.
   OR
   Click the **Chart Options** button. The **JClass Chart Properties** dialog box is displayed.
2. On the **JClass Chart Properties** dialog box, make the necessary changes to the appearance of the container vitals graph.

Note: Visit the Sitraka website at **//www.sitraka.com** for documentation on the JClass Chart Customizer.

Graph performance vitals

To graph performance vitals for a component:

1. Select **Component >Show Performance Vitals**.
   OR
   On the **Graphing Components** window, click the **Show Performance Vitals** button. The **Select Component to View Performance** dialog box is displayed. This dialog box lists all the available components you can view performance vitals for.
2. Select the component you wish to view performance vitals for. The selected component is highlighted in blue.
3. Click **OK**. The **Graphing Performance** window is displayed. The performance graph is empty.
4. Select **Series** and then the appropriate performance vital you wish to view. The performance graph displays the performance statistics for the selected vital.

Note: By default, all methods are selected when you first display the **Graphing Performance** window. The performance graph will display performance statistics for all methods within the selected component or service.

View multiple performance vitals

You can view multiple performance vitals simultaneously. Performance vitals are color coded so you can easily distinguish between performance vitals. Each time you view a performance vital, the graph legend displays the selected performance vitals and the assigned color.

To view multiple component vitals simultaneously:

1. On the **Graphing Performance** window, select **Series** and the appropriate menu option for the vital you wish to view. A bar graph representing the vital is displayed on the vitals graph.
2. To display multiple performance vitals simultaneously, repeat step 1. A new bar graph representing the selected vital is displayed on the vitals graph along with the previously selected vitals.

Deselect performance vitals

After you have displayed multiple performance vitals for a component, you can deselect a performance vital so it is no longer displayed on the **Graphing Performance** window.

To deselect a performance vital:

1. Select **Series** and the appropriate previously selected menu option to deselect a performance vital. The performance vital is no longer displayed on the performance graph.
2. To deselect more performance vitals so they are no longer displayed on the performance graph, repeat Step 1.

Deselect a method

By default, the **Graphing Performance** window displays performance vitals for all methods within a component. Mission Control allows you to customize which methods will be displayed on the performance graph. You to deselect a method so it no longer is displayed on the performance graph.

To deselect a method:

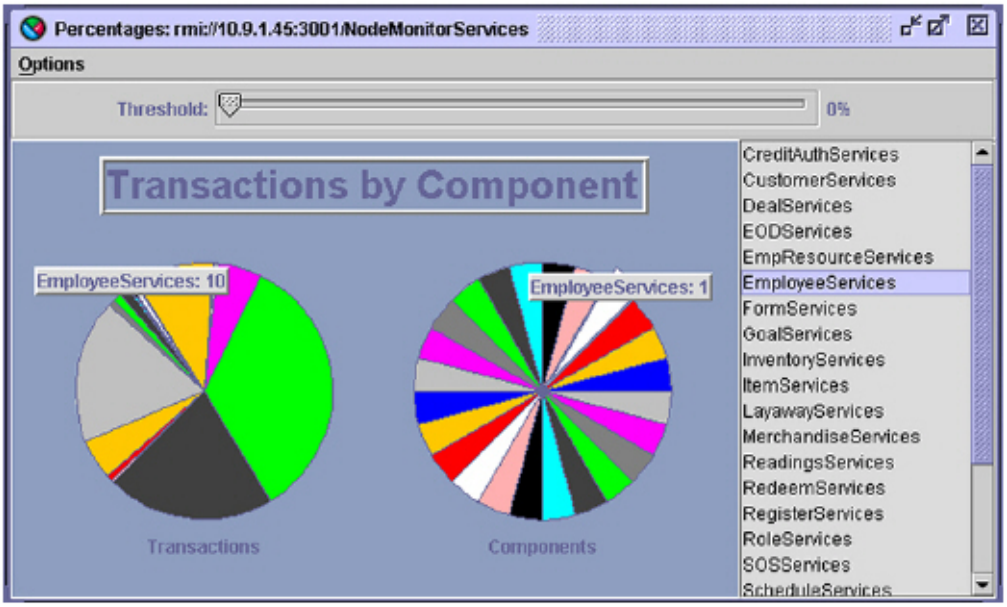
1. On the **Graphing Performance** window, select **Methods>Show Methods**. A submenu with all the available methods is displayed. All listed methods are selected with a check mark.
2. On the submenu, select the method you wish to no longer view the performance vitals for. Performance statistics for the selected method are no longer displayed on the performance graph.
3. To deselect multiple methods, repeat steps 1-2.

View percentage charts

The **Percentage Chart** features allows you to view and monitor the workload of each container and component. The percentage charts display the number of transactions per container or component so you can balance the workload for optimal performance levels.

Percentages window overview

The **Percentages** window shows the number of instances of a container or component compared to the number of transaction performed by a container or component. Each of these numbers is displayed in pie charts that appear side-by-side for comparison purposes. This view allows you to increase and decrease the percentage used by a component to match the percentages of the transactions performed. Ideally in a production environment, the percentages chart for transactions will match the percentage charts for either the containers or components.



Percentages window

The Percentages window contains one menu item, the **Options** menu, and it does not have a toolbar or command buttons. The **Options** menu contains the following menu options:

- **View by Container**—Allows you to view the **Transactions by Container** chart. This is the default setting.
- **View by Component**—Allows you to view the **Transaction by Component** chart.
- **Chart Options**—Allows you to customize the appearance of the percentage charts.

Viewing options

The viewing options on the **Percentages** window allow you to customize the percentages chart and the information displayed on the **Percentages** window. The table below lists and describes the viewing options.

Percentages Window Viewing Options	
Viewing Option	**Description**
Threshold slider	Allows you to customize the appearance of the chart by increasing or decreasing the threshold value. The threshold value is the percent of transactions a component performed during the time performance statistics were collected. Increasing or decreasing the threshold value along the slider will either increase or decrease the number of pieces that appear on the chart.
Containers and Containers list	Allows you to display a label over the corresponding piece of the pie chart. Multiple labels can be viewed at one time. The containers list displays the name of the connected container. The components list displays all the components within the connected container.
Labels	Displays the container name, component name, and number of transactions that the container or component performed over the corresponding piece of the pie chart.

View percentage charts

To view the percentage charts:

1. Connect to an application server. The **Containers** window for that application server is displayed.
2. On the **Containers** window, select the container you wish to view the vitals for. The selected container is highlighted in blue.
3. Select **Graph**>**Percentages**.
   OR
   Click the **Chart Percentages** button. The **Percentages** window is displayed.

Note: By default, the **Percentages** window displays the **Transaction by Container** graph.

Change percentage chart

By default, the **Percentages** window displays the **Transaction by Container** chart when you first connect to the **Percentages** window. The **Percentages** window allows you to view the following percentage charts:

- View by Container
- View by Component

To change the displayed percentage chart:

- On the **Percentages** window, select **Options** and then the appropriate menu option for the percentage chart you wish to view. For example, if you wish to view the percentage chart for a component, select **Options>View by Component**. The selected percentage chart is displayed on the **Percentages** window.

Modify the threshold percentage

The threshold slider allows you to customize the appearance of the chart by increasing or decreasing the threshold value. The threshold value is the percent of transactions a component performed during the time performance statistics were collected. Increasing or decreasing the threshold value along the slider will either increase or decrease the number of pieces that appear on the chart.

To modify the threshold percentage:

- Move threshold slider to the desired percentage value. If the threshold percentage is greater than the threshold percentage of a container or component, the corresponding piece of the pie chart disappears. The piece of the pie chart is added to a combined piece that represents all containers or components that have a threshold percentage less than the chosen threshold percentage.

Display labels

Mission Control allows you to display labels for each piece of the percentages chart. The labels display the following information for a container and component:

- Name
- Number of instances
- Number of transactions performed by the container or component

When you display a label, the label appears on the percentage chart for both the percentage and container or component, depending on which view you have displayed. This allows you to see the relationship between the number of transactions a container or component is performing to the number of instances of a container or component. With this information, you can make decisions regarding the workload of a container or component. Also, you can display multiple labels at one time.

To display a label:

1. Select the name of the container or component from the container or component list.
   The selected container or component is highlighted in blue and the corresponding label is displayed.
   OR
   Click the desired piece of the percentage chart in the transactions chart. The corresponding container or component is highlighted in the container or component list.
   OR
   Click the desired piece of the percentage chart in the container or components chart. The corresponding container or component is highlighted in the container or component list.
2. To display multiple labels, repeat step 1.

Design options

The **Options** button on the **Percentages** window allows you to customize the appearance of the performance charts and the **Percentages** window itself. The **Options** button starts Sitraka's JClass Chart Customizer that allows you to customize and view the graph properties.

To customize the graph properties:

1. Select the **Options** button.
2. Select from the various options.

Note: Visit the Sitraka website at //www.sitraka.com/ for documentation on JClass Chart Customizer.

Log files overview

Mission Control allows you to view the following log files:

- **Containers** log file—Allows you to view system out files, system error files, and event logs.
- **Components** log file—Allows you to view log files that can be filtered by levels.
- **Clients** log file—Allows you to view log files, system out files, and system error files.

When you connect to a application server or client to view log files, the **View Log File** window is displayed. The viewing options in the **View Log File** window varies for containers, components, and clients.

Container log files

The table below provides the information and options displayed on the **View Log File** window for containers.

View Log File Window Properties for Containers	
Window Property	**Description**
UID	Provides the unique ID for the container you are viewing log files for.
System Out radio button	Allows you to view system out files.
System Error radio button	Allows you to view system error files.
Events radio button	Allows you to view event logs.
Refresh button	Allows you to refresh the View Log File window to display either an updated log file or a new log file.
Clear System Error button	Allows you to clear all error files from a container.
Save button	Allows you to save log files as a text file.
Close button	Allows you to close the View Log File window.

View container log files

To view container log files:

1. Connect to an application server. The **Containers** window is displayed for the connected server.
2. Select the container you view to view log files for. The selected container is highlighted in blue.
3. Select **Container>View Logs**.
   OR
   Click the **Logging** button. The **View Log Files** window is displayed.

Note: If you do not select a container, the Select Container dialog box is displayed.

4. Choose the type of log file you wish to view. By default, the **Events** option is selected.
5. Click the **Refresh** button. The log files for the selected option is displayed.
6. To view a different log file, repeat steps 4 and 5.
   OR
   To refresh the current log files, click the **Refresh** button.

Clear the system error files

The **Clear System Error** option allows you to clear all system error files for the selected container. When a container contains a system error, the **Exception** button is displayed in red. After you clear all system errors, the **Exception** button is displayed in green.

To clear the all system error files for a container:

1. View the log files for the container that contains the exception error.
2. On the **View Log Files** window, click the **Clear System Error** button. The **Clear Error Log** dialog box is displayed.
3. On the **Clear Error Log** dialog box, click **Yes**. The **Error Log Cleared** dialog box is displayed.
4. On the **Error Log Cleared** dialog box, click **OK**.

Save container log files

Mission Control allows you to save container log file as text files. You can save these files in a user-defined directory.

To save container log files:

1. View the log files for the type of log files you wish to save.
2. Click the **Save** button. The **Save** dialog box is displayed.
3. Select the directory you wish to save the log files in.
4. On the **Save** dialog box, click the **Save** button. The selected log file is saved as a text file in the defined directory.

Component log files

The table below provides the information and options displayed on the **View Log File** window for components.

View Log File Window Properties for Components	
Window Property	**Description**
Entries Retrieved	Allows you to control the number of entries to display.
Log File Type drop-down menu	Allows you to view a type of log file.
Refresh button	Allows you to refresh the View Log File window to display either an updated log file or a new log file.
Save button	Allows you to save log files as a text file.
Close button	Allows you to close the View Log File window.

View component log files

Mission Control allows you to customize your log file properties using the **Log File Type** drop-down menu. The types of log files are divided into hierarchical levels of severity. There are four levels of log files that correspond with the information contained within them. The following are the four different types of log files:

- **Informational**—Stores all the information logged by the system, including minor, major, and fatal errors and exceptions.
- **Minor**—Stores all minor, major, and fatal errors and exceptions.
- **Major**—Stores all major and fatal errors.
- **Fatal**—Stores all fatal errors only.

When viewing a type of log file, Mission Control displays not only the selected type of log file, but all types of log files above it in the hierarchy. For example, if you choose to view all **Major** execption errors, Mission Control will display both **Major** and **Critical** execption errors.

To view component log files:

1. Connect to an application server. The **Containers** window is displayed for the connected server.
2. Select the component you view to view log files for. The selected component is highlighted in blue.
3. Select **Component>View Logs**.
   OR
   Click the **Logging** button. The **View Log Files** window is displayed.

Note: If you do not select a component, the **Select Component** dialog box is displayed. Click **OK** to close the dialog box.

Note: If there are no log file entries, the **No Log File Entries** dialog box is displayed.

4. Click the **Log File Type** drop-down menu and select the type of log file you wish to view. By default, the **Info** option is selected.
5. Click the **Refresh** button. The log files for the selected type of log file are displayed on the **View Log Files** window.

Modify number of entries retrieved

Mission Control allows you to modify the number of entries retrieved and displayed on the **View Log Files** window. To modify the number of entried retrieved and displayed:

1. On the **View Log Files** window, enter the maximum number o fentried you wish to retrieve.
2. Click the **Refresh** button. Mission control refreshes the **View Log Files** window with the most recent error files.

Change the type of log file displayed

By default, the **View Log Files** window displays all four types of log files. You can modify the type of log file displayed on the **View Log Files**. Remember, Mission Control displays not only the selected type of log file, but all types of log files above it in the hierarchy.

To change the type of log file displayed:

1. Click the **Log File Type** drop-down menu and select the type of log file you wish to view.
2. Click the **Refresh** button. The log files for the selected type of log file are displayed on the **View Log Files** window.
3. To view additional log file types, repeat steps 1-2.

Save container log files

Mission Control allows you to save component log file as text files. You can save these files in a user-defined directory.

To save component log files:

1. View the log files for the type of log files you wish to save.
2. Click the **Save** button. The **Save** dialog box is displayed.
3. Select the directory you wish to save the log files in.
4. On the **Save** dialog box, click the **Save** button. The selected log file is saved as a text file in the defined directory.

Chapter 8 – Message monitor

Retek® RPOS™ Tools Mission Control allows you to publish (send) messages to clients and server-side components using a Java Messaging Service (JMS) Server. Mission Control employs the publish/subscribe messaging model in which messages are published to and retrieved from named topics.

Using Message Monitor, you can view your network topology. The network topology allows you to view representations of each RPOS process, including IP addresses, configuration ports, and exceptions that occur with each process. By expanding and collapsing the view of the network topology, you can drill down all the way to a client register running in a store. While viewing the network topology, Message Monitor allows you to connect to any node on the network without having to enter an IP address or port number.

Message Monitor allows you to perform the following tasks:

- View the network topology in a hierarchical view of IP addresses
- Connect to a node (client, naming server, or application server) on the network
- Publish messages
- View exception errors and start RPOS Tools Object Inspector

Connect to message monitor

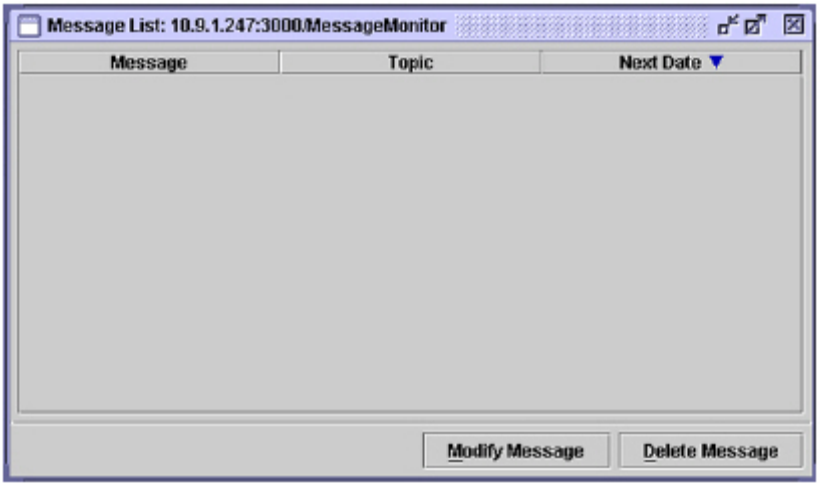
You can connect to Message Monitor the same way you would any node on the network. Before you can connect to Message Monitor, you will have to set up Message Monitor using the **Monitor** feature. See Chapter 2, “Set Up a Node,” for more information on setting up and connecting to Message Monitor.

After you connect to Message Monitor, the **Message List** window and the **Message Monitor** window are displayed. The **Message List** window is displayed within the **Mission Control** workspace while the **Message Monitor** window is displayed as a separate window from the **Mission Control** workspace.

Message list window overview

The **Message List** window is displayed within the **Mission Control** workspace after you connect to Message Monitor. The **Message List** window displays the following information:

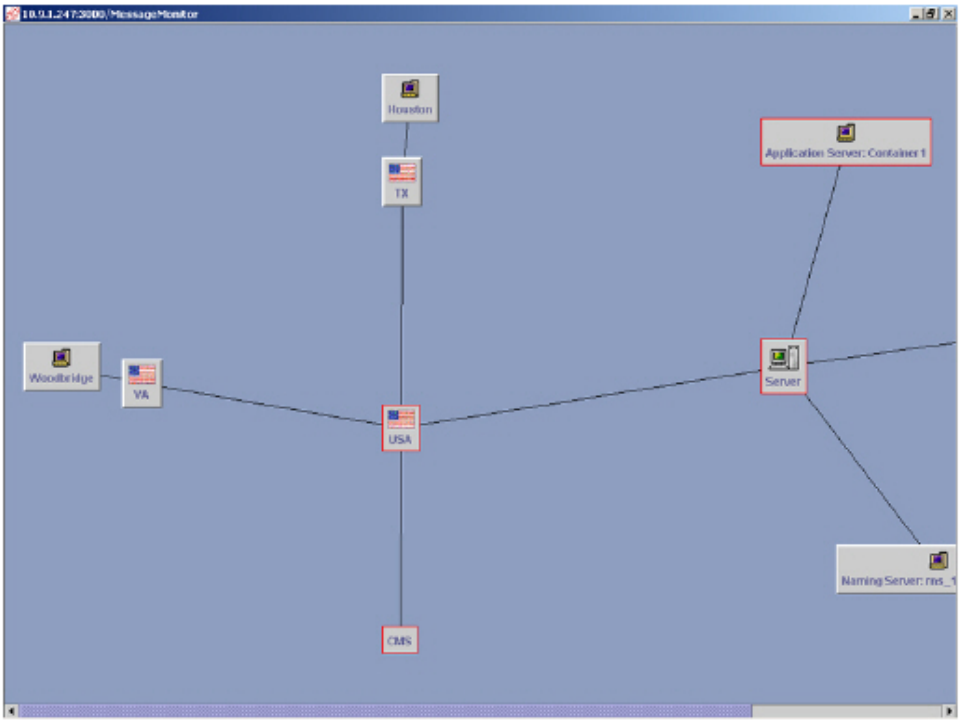
- **Message**—Provides the names of the published messages.
- **Topic**—Provides the topics of the published message.
- **Next Date**—Provides the dates in which the messages were published.
- **Modify Message** button—Allows you to modify published messages by opening the **Message Wizard**.
- **Delete Message** button—Allows you to delete published messages.



Message List window

Message monitor window overview

The **Message Monitor** window is displayed as a separate window from the **Mission Control** workspace. The Message Monitor window displays the network topology that allows you to drill down and view the different levels and nodes on the network, connect remotely to nodes, assign icons to nodes, view exceptions, and launch Object Inspector.



Message Monitor window

View network topology

Message Monitor provides a hierarchical view of the RPOS clients and servers in your network. The hierarchy is based on a grouping of stores and server-side components. The grouping structure is configurable. The root node is generally the company name and all other nodes branch out from that root node.

In the RPOS system, a store object knows which group it belongs to, and the clients or registers within the store are automatically grouped to that store. The message grouping is used to target messages to specific groups.

Message Monitor allows you to view the network topology based on your needs. You can view the network from something as general as the different countries within the network or you can drill down to an actual store client in New York City.

To view the network topology:

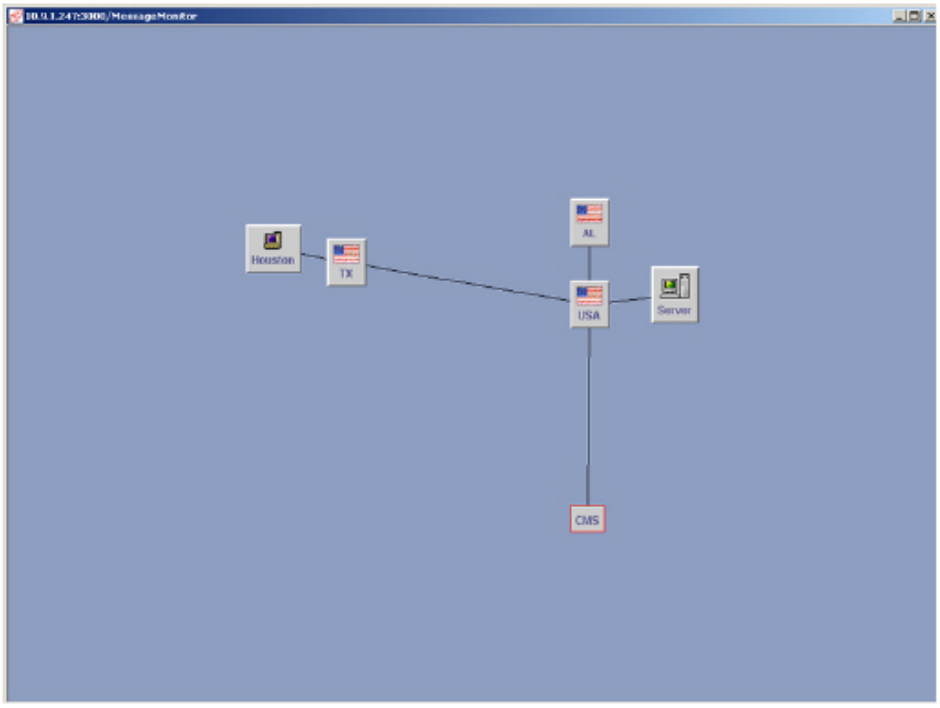
- Connect to Message Monitor. The **Message Monitor** window is displayed as a separate window from the **Mission Control** workspace.

Note: The view of your network topology may be different each time you connect to Message Monitor. The viewable nodes in the **Message Monitor** window are all nodes on the same level as the node last doubled clicked during the last session of Mission Control and Message Monitor. For example, the last time you used Message Monitor, you viewed a specific client in your Houston store #115. The next time you use Message Monitor, the viewable nodes in the **Message Monitor** window will be the store level because that was the last node you double clicked to expand.

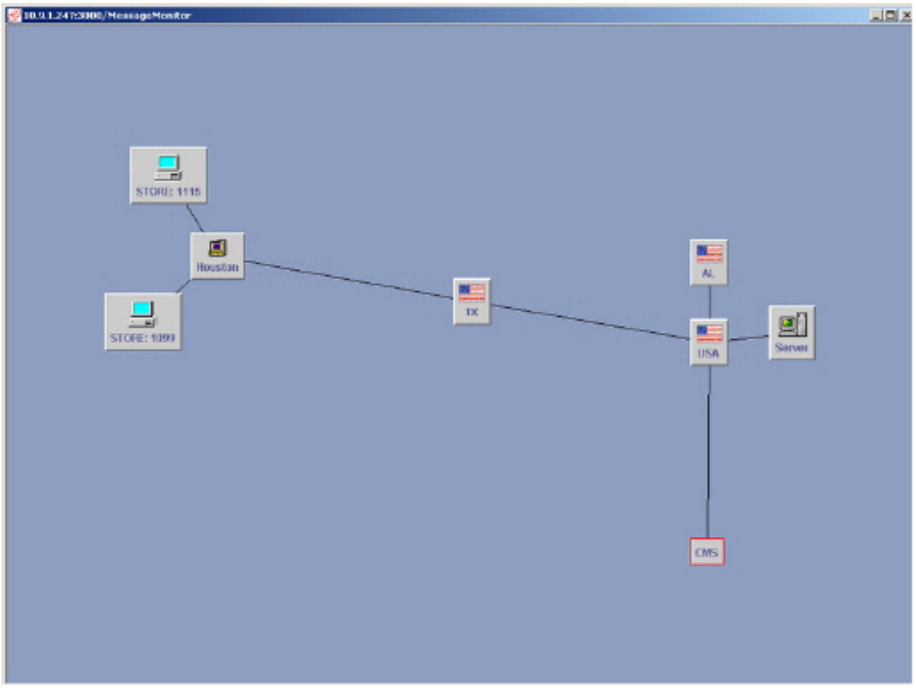
Expand a node or level

In the RPOS system, Message Monitor allows you to expand the network topology and drill down to the client level. Expanding a node does not expand all nodes on that level, just the selected node. To expand a node or level on the network topology:

1. Connect to Message Monitor. The **Message Monitor** window is displayed with a view of the network topology.



2. Double click on the node or level you wish to expand. An expanded view of the network topology is displayed, and you can view the next level on the network.



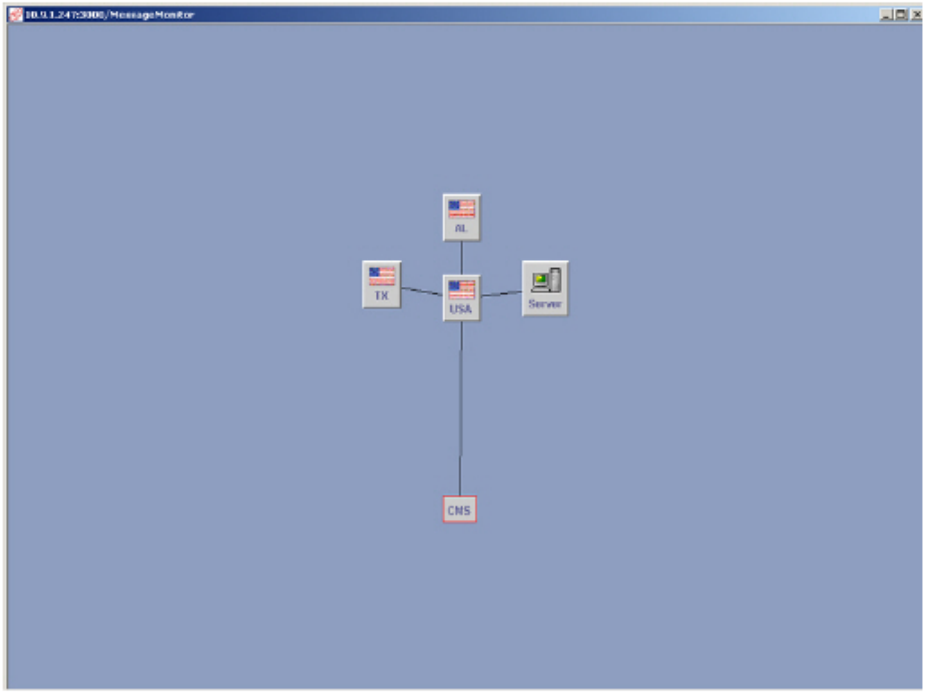
3. To drill down and expand the network topology to display more nodes or levels on the network, repeat Step 1.

Collapse a node or level

Message Monitor also allows you to collapse a node or level on the network. Collapsing a node does not collapse all nodes on that level, just the selected node. To collapse a level, you must double click on the grandparent level of the level you wish to collapse. For example, your network is grouped into a state level, a store level, and a client level. You have expanded the network topology to view the client level but now you are ready to collapse that level and no longer view the client level. Double clicking on the client or the store level will not collapse the client level. You must double click the grandparent level, and in this case, the state level.

To collapse a specific level on the network topology:

1. Double click on the grandparent level of the level you wish to collapse. The network topology will collapse, and you can no longer view that level.



2. Repeat Step 1 to collapse another level.

To collapse the entire network topology:

- Double click on the root node of the network topology. The network topology will collapse to display only the root node and the subsequent level.

Assign icons to nodes

You can assign icons to nodes on the network. Assigning icons to nodes helps you view and identify different nodes and levels on the network. The icon assigned to node can be any image file, such as a GIF or JPEG file. The following are two ways you can assign icons to nodes and levels:

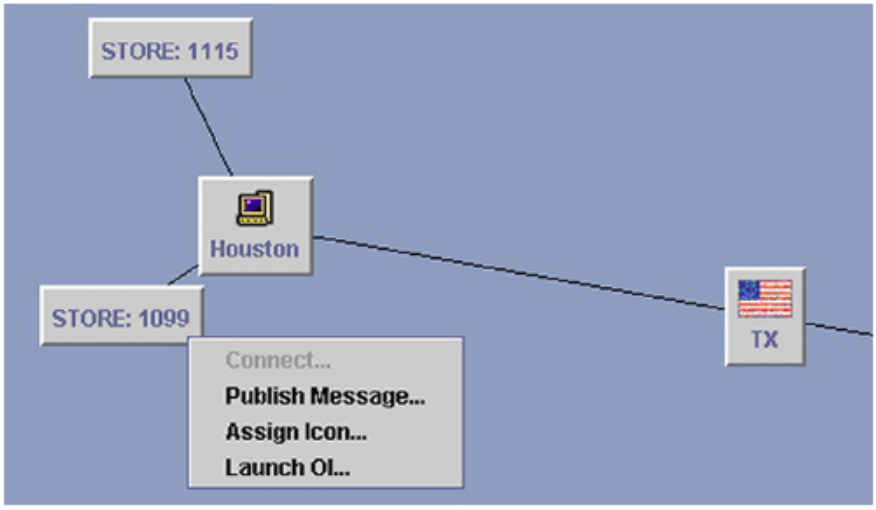
- Assign an icon to all nodes on a level
- Assign an icon to all nodes with this name

Assign an icon to all nodes on a level

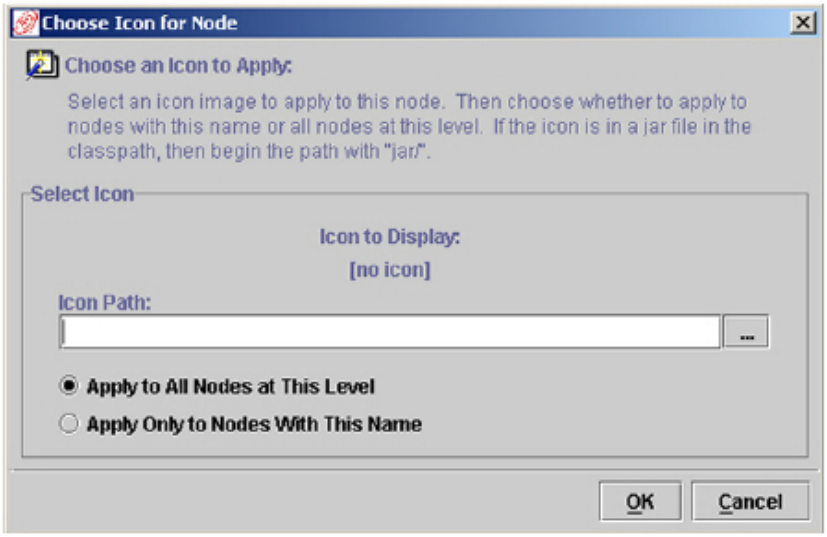
Mission Control and Message Monitor allow you to assign icons to all nodes on a selected level. This feature allows you to create a uniform network-wide appearance for all nodes on a specific level. For example, you can group your network into countries, states or provinces, cities, stores, and client terminals. You can assign a different icon to represent city, store, and client levels. A separate icon for each node or level helps distinguish that level from the other levels on the network.

To assign an icon to all nodes on a level:

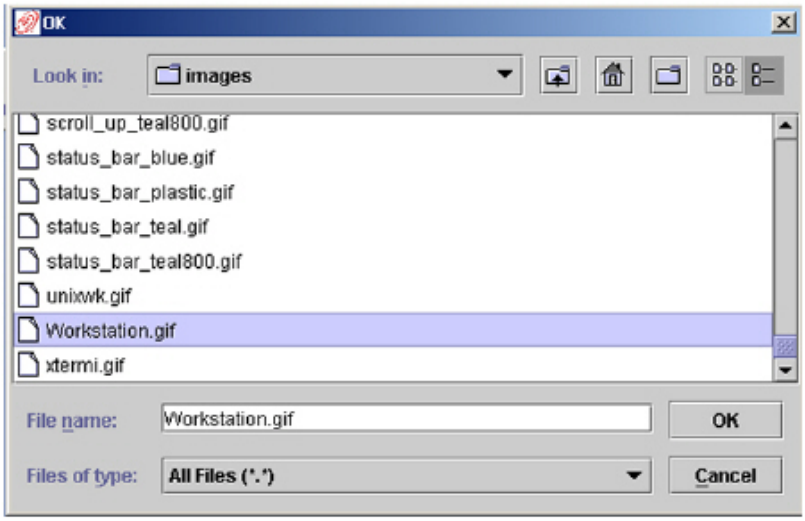
1. Connect to Message Monitor. The **Message Monitor** window is displayed with a view of the network topology.
2. Right click on a node on the network level you wish to assign the icon to level-wide. A pop-up list is displayed.



3. In the pop-up list, select **Assign Icon**. The **Choose Icon for Node** dialog box is displayed.

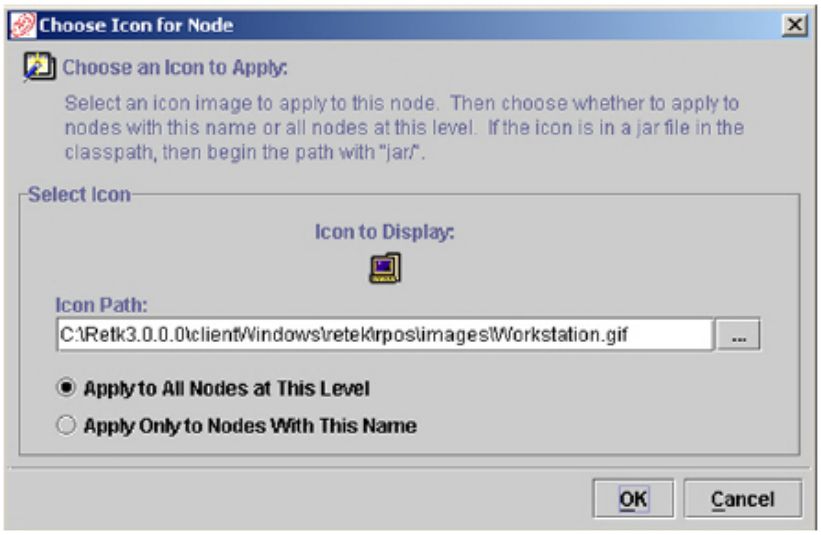


4. To select an image file that is located in a directory:
   a. Click the **Find Icon** button. The **Directory** dialog box is displayed.
   b. In the **Directory** dialog box, locate and select the desired image file. The selected image file is highlighted in blue.

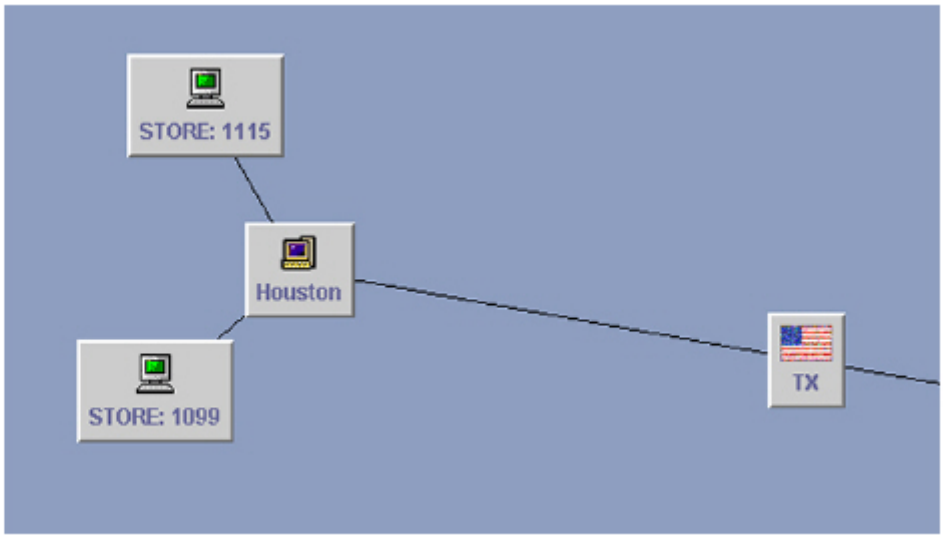


   c. Click **OK**. The selected image file is displayed in the **Choose Icon for Node** dialog box.
   OR
   To select an image file that is located in a JAR file in the classpath:
      a. In the **Icon Path** field, type the classpath for the image file. Begin the classpath by typing `jar/` and then the rest of the classpath.
      b. After you have typed the classpath for the image file, press **Enter**. The selected image file is displayed in the **Choose Icon for Node** dialog box.

5. In the **Choose Icon for Node** dialog box, verify that the correct icon has been selected.



6. Select the **Apply to All Nodes on This Level** checkbox.
7. Click **OK**. The selected icon is displayed for all nodes on the selected level.

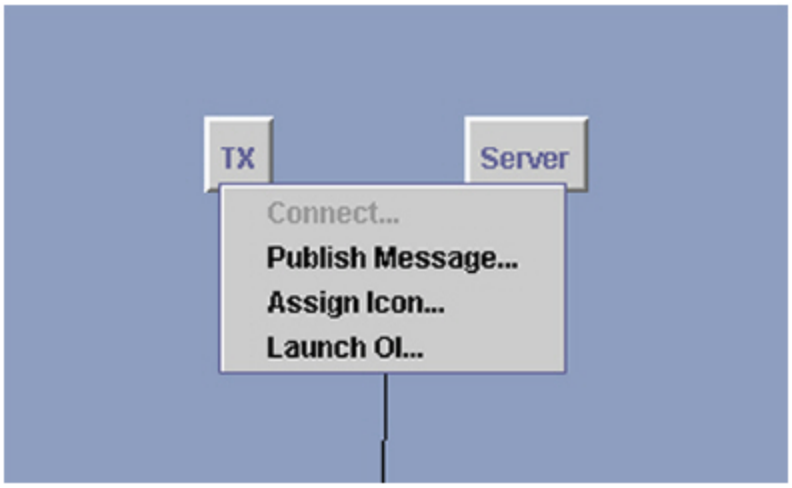


Assign an Icon to Only Nodes with the Same Name

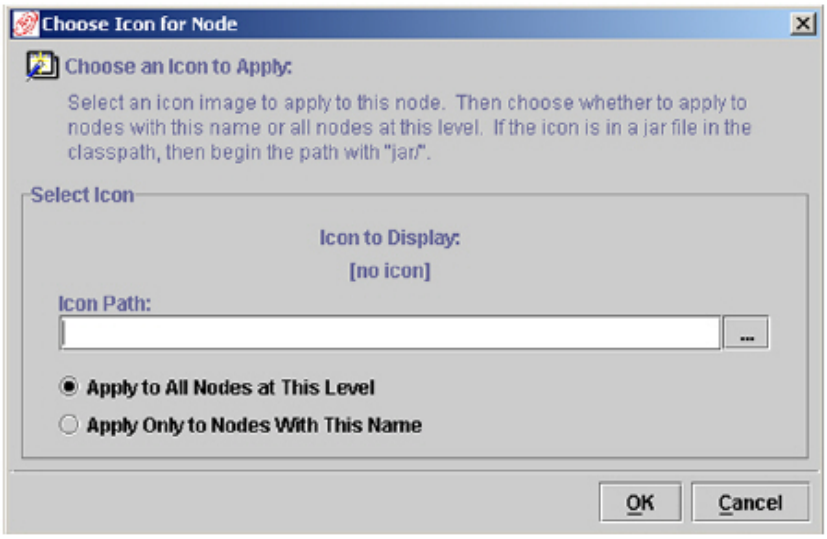
Mission Control and Message Monitor allows you to assign icons to only nodes or levels with the same name. This feature allows you to localize a icon for a specific node or level. For example, you set up Message Monitor to group your network by countries, states or provinces, cities, stores, and then client terminals. If you have stores in the United States, Canada, and Mexico, you can use each countries' flag to represent that particular node. Also, you can use the state flag for each state to help you distinguish the different nodes on that particular network level.

To assign an icon to only nodes with the same name:

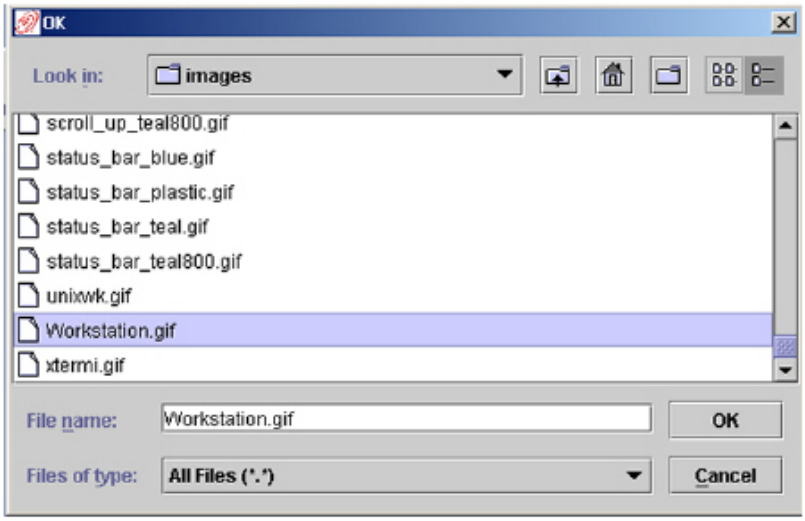
1. Connect to Message Monitor. The **Message Monitor** window is displayed with a view of the network topology.
2. Right click on a node on the network level you wish to assign the icon to level-wide. A pop-up list is displayed.



3. In the pop-up list, select **Assign Icon**. The **Choose Icon for Node** dialog box is displayed.

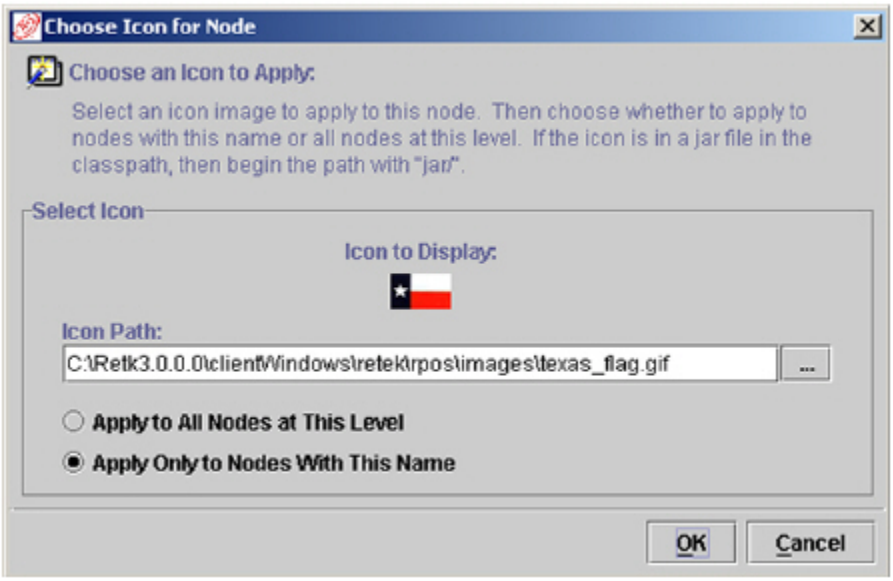


4. To select an image file that is located in a directory:
   a. Click the **Find Icon** button. The **Directory** dialog box is displayed.

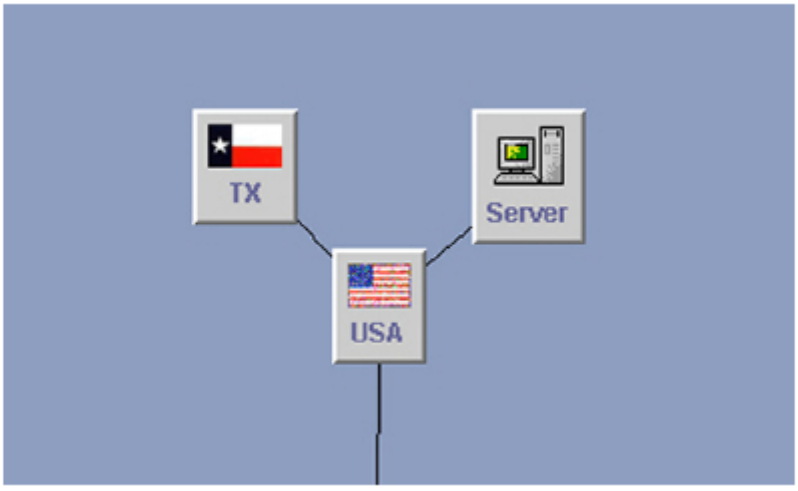


   b. In the **Directory** dialog box, locate and select the desired image file. The selected image file is highlighted in blue.
   c. Click **OK**. The selected image file is displayed in the **Choose Icon for Node** dialog box.
   OR
   To select an image file that is located in a JAR file in the classpath:
      a. In the **Icon Path** field, type the classpath for the image file. Begin the classpath by typing `jar/` and then the rest of the classpath.
      b. After you have typed the classpath for the image file, press **Enter**. The selected image file is displayed in the **Choose Icon for Node** dialog box.

5. In the **Choose Icon for Node** dialog box, verify that correct icon has been selected.



6. Select the **Apply Only to Nodes With This Name** checkbox.
7. Click **OK**. The selected icon is displayed on all nodes with the same name as the selected node.



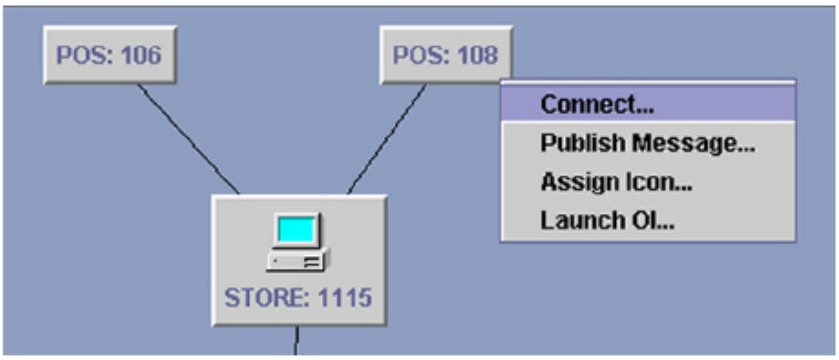
Connect to a node

Using Message Monitor, you can connect to any node or device on the network without having to enter an IP address or port number for a node. Connecting to a node allows you to view, manage, and monitor that node from a remote location.

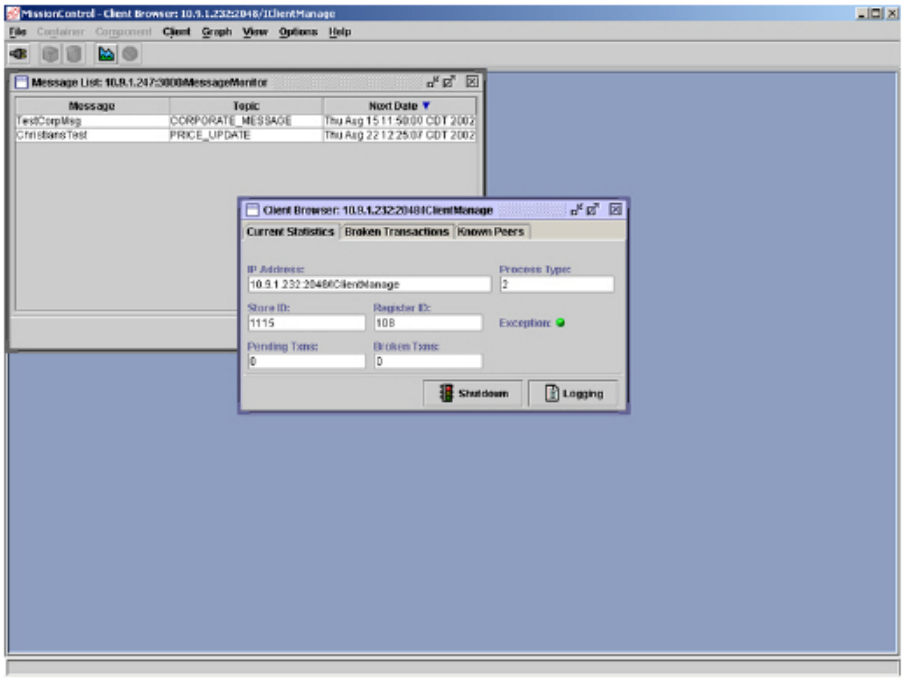
After you connect to a node using Message Monitor, the appropriate window is displayed in the **Mission Control** workspace. For example, connecting to a client using Message Monitor will display the **Client Browser** window for the selected client. You can connect to multiple nodes at one time.

To connect to a node using Message Monitor:

1. Connect to Message Monitor. The **Message Monitor** window is displayed with a view of the network topology.
2. Drill down to the network level that contains the node you wish to connect to.
3. Right click on a node you wish to connect to. A pop-up list is displayed.



4. In the pop-up list, select **Connect**. In the **Mission Control** workspace, the appropriate window is displayed depending on the type of node you connected to.



Publish a message

Message Monitor allows you to send messages to specific message groups. For example, you can send a message to the Dallas, Texas group that will be received by all nodes and devices subscribed to the Dallas, Texas group. This message will not be received by the Houston, Texas group. However, if you send a message to the entire Texas group, this message will be received by all nodes and devices subscribed to the Texas group, including the Dallas and Houston groups.

Mission Control employs the **Message Wizard** to assist you in creating and publishing a message. To create a message using **Message Wizard**, you will need to provide the following information:

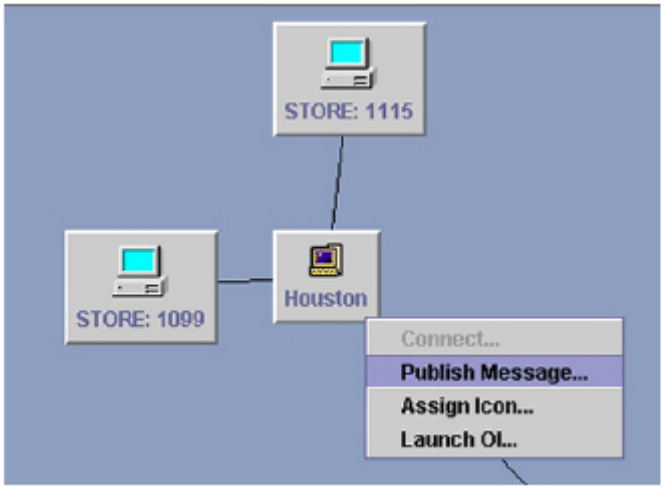
- **Message name**—It is recommended you select a name that describes the purpose or contents of the message. This name will not be seen by receivers of the message, but it is used to save the message to be sent again later.
- **Group name**—The group name is the name of the group that will receive this message. The default group name is the node group of the node you right clicked. However, you can change the group name using the **Message Wizard**.
- **Topic key**—The topic key is the topic that subscribers will listen for to receive this message. Only nodes within the group name that are subscribed to this topic will receive this message. You select a topic from a drop-down menu.
- **Message properties**—You will build the contents of the message by adding key-value pairs. The key-value pairs can be instructions for a subscriber to receive information or perform a task. For example, if you are sending a corporate message, you will need to add a key-value pair for the message itself and the start and stop date of the message.
- **Desired action**—The message can be schedule as a one-time event, regular event, or schedule with a specific JavaSchedule Java object.

- **JobSchedule classname**—You may have to build a JobSchedule JavaClass by specifying a classname.

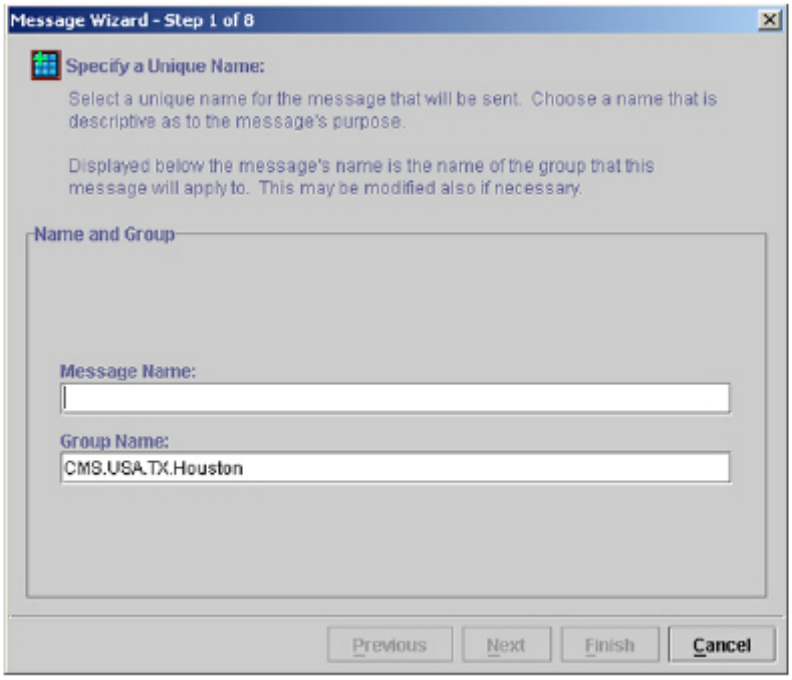
After you have created the message, you will have to confirm the contents and actions of the message. **Message Wizard** allows you to view the message properties at once so you can either send the message or make the necessary changes.

To publish a message:

1. Connect to Message Monitor. The **Message Monitor** window is displayed with a view of the network topology.
2. Right click on the parent node you wish to publish the message to. A pop-up list is displayed.

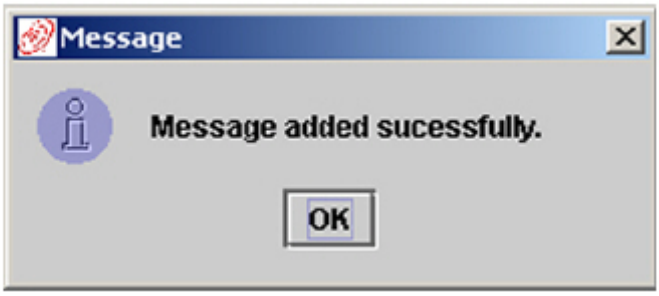


3. In the pop-up list, select **Publish Message**. The **Message Wizard** is displayed.

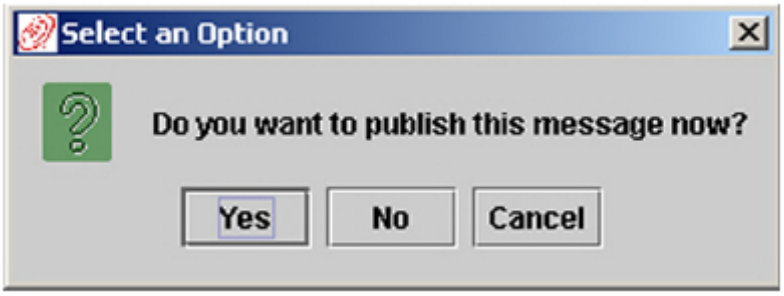


4. Follow the eight steps within the **Message Wizard** to successfully create and send a message using Message Monitor.

5. After you have created a new message, click the **Finish** button on the **Message Wizard**. The **Message Posted** dialog box is displayed, and the message information is displayed in the **Message List** window in the **Mission Control** workspace.



6. In the **Message Posted** dialog box, click **OK**. The **Post Message Now** dialog box is displayed.



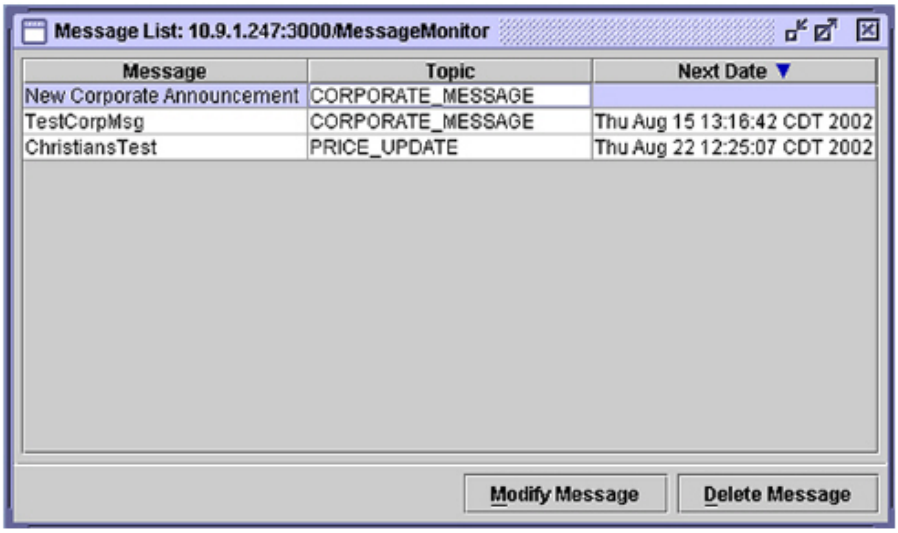
7. If you want to post the message now, click **Yes**.
   OR
   If you want to post the message at another time, click **No** or the **Cancel** button.
8. To close **Message Wizard**, click **OK**.

Modify a message

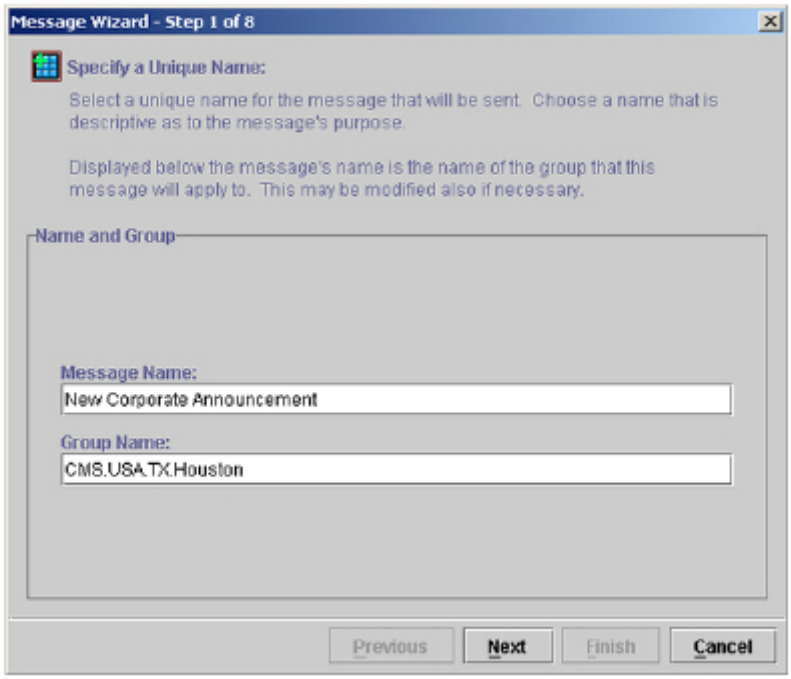
Mission Control allows you to modify an existing message from the **Message List** window. The **Message Wizard** is displayed so you can make modifications to the properties of the message.

To modify a message:

1. Connect to Message Monitor. The **Message List** window is displayed in the **Mission Control** workspace.
2. In the **Message List** window, select the message you wish to modify. The selected message is highlighted in blue.



3. Click the **Modify Message** button. The **Message Wizard** is displayed in the **Message Monitor** window.



4. Using the **Message Wizard**, locate the message property you wish to change.

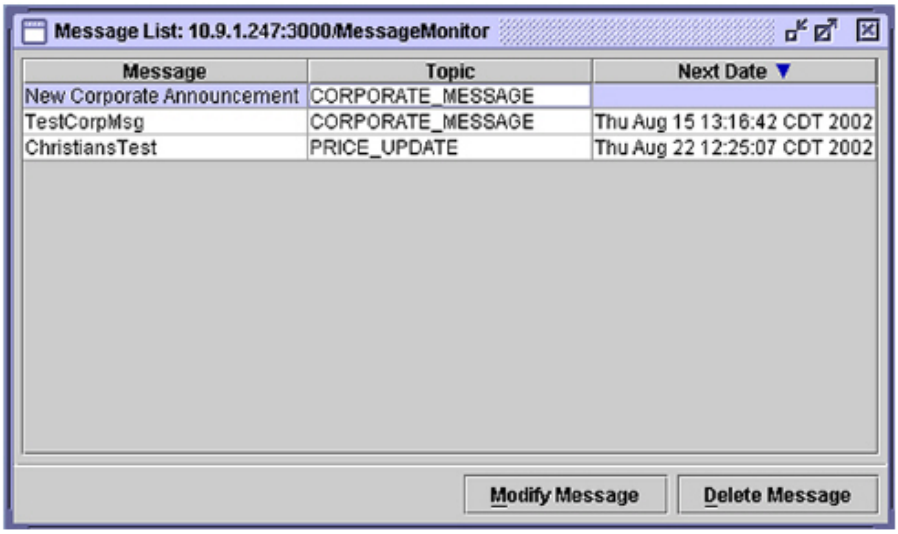
   Note: To navigate through the **Message Wizard**, click the **Next** button.

5. Make the necessary changes to the message.
6. After you have made all your changes, click the **Finish** button. The **Message Wizard** is no longer displayed, and your changes to the message are saved.

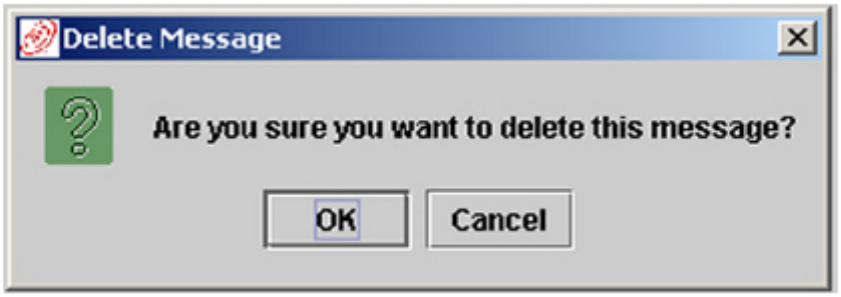
Delete a message

Mission Control allows you to delete a message. To delete a message:

1. Connect to Message Monitor. The **Message List** window is displayed in the **Mission Control** workspace.
2. In the **Message List** window, select the message you wish to delete. The selected message is highlighted in blue.



3. Click the **Delete Message** button. The **Delete Message** dialog box is displayed.



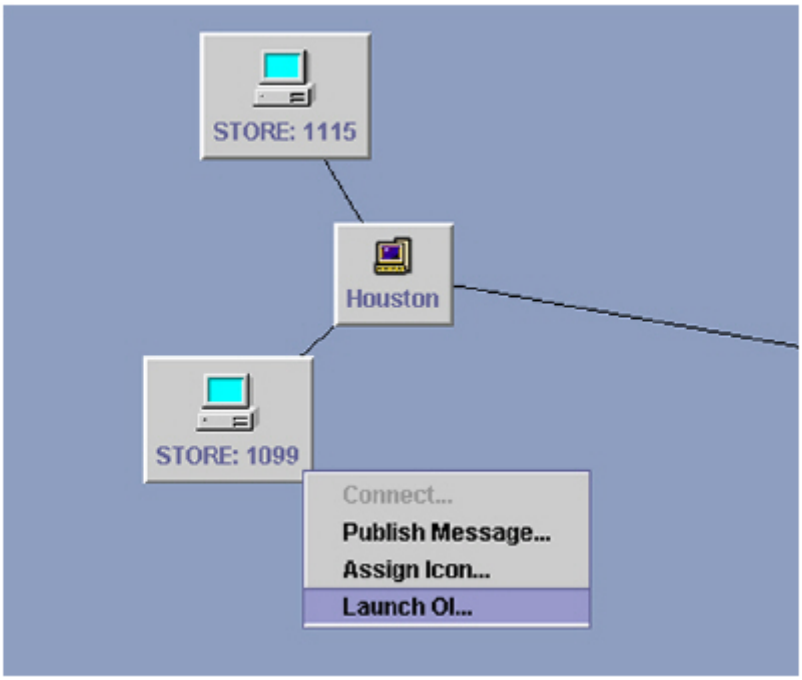
4. Click **OK**. The message is no longer displayed in the **Message List** window.

View broken transactions in RPOS tools object inspector

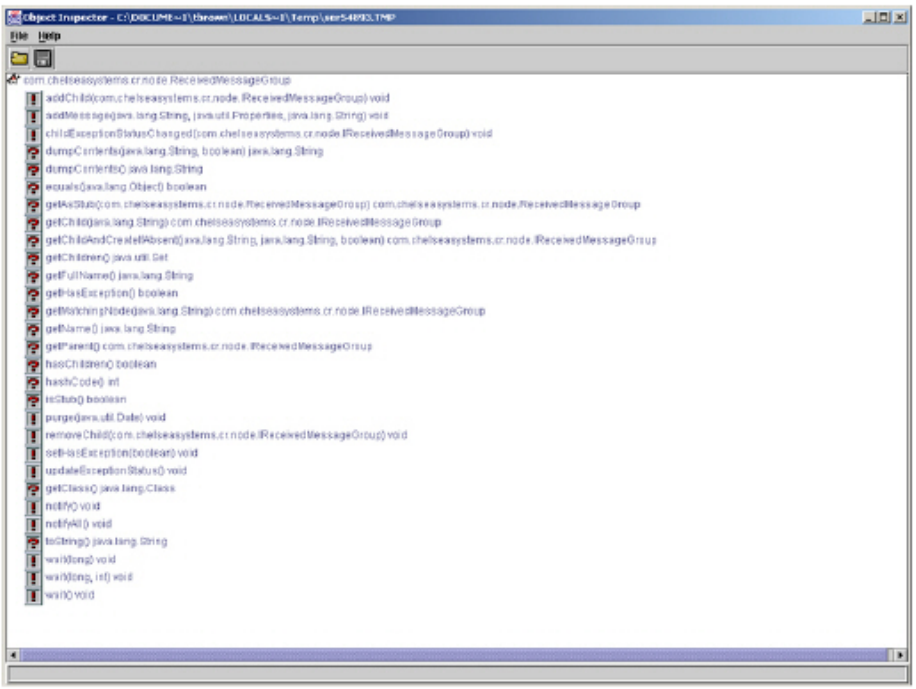
Message Monitor allows you to view the object that represents that particular node in RPOS Tools Object Inspector. If a node contains an exception error, the parent level is displayed with a red border in Message Monitor. You will have to drill down to the node and examine the exception in Object Inspector.

To view a broken transaction in Object Inspector:

1. Connect to Message Monitor. The **Message Monitor** window is displayed with a view of the network topology.
2. Right click on the node that contains the exception. A pop-up list is displayed.



3. In the pop-up list, select **Launch OI**. Object Inspector is started in a separate window from both the **Mission Control** workspace and the **Message Monitor** window.



Note: For more information on how to view or fix broken transaction using Object Inspector, see the **RPOS Tools Object Inspector User Guide**.

Chapter 9 – Manifest process

Software updates and data file updates are pushed out from the application server in an automated process called a manifest. The manifest can contain any combination of files, including the following: software updates, images, binary files, text files, JVMs, PLU updates, and so on. The process uses a combination of TCP/IP and UPD/IP for peer-to-peer (P2P) pushes of the manifest between client terminals.

There are two types of manifest updates: full and incremental. A manifest is pushed out to the clients in a two-phase process. Each client is scheduled to check for updates at a pre-scheduled time determined by the system administrator.

If there is more than one client in a store, the first client that receives the manifest will lock out all other clients in that location from receiving the manifest. Once the first client receives the manifest, it uses P2P networking to update the remaining in-store clients. The initial push sends the updates in a "wait-to-install" state that allows system administrators time to check the manifest logs and verify that all clients receive the updates and that there are no network or hardware outages.

You can change the manifest status to install on the application server, or the update is completely removed. When each client checks in at a pre-scheduled time and sees the status as "install," the client installs the new software or data files. The system administrator can now view the manifest logs to determine if there are any errors or clients that did not install the manifest properly.

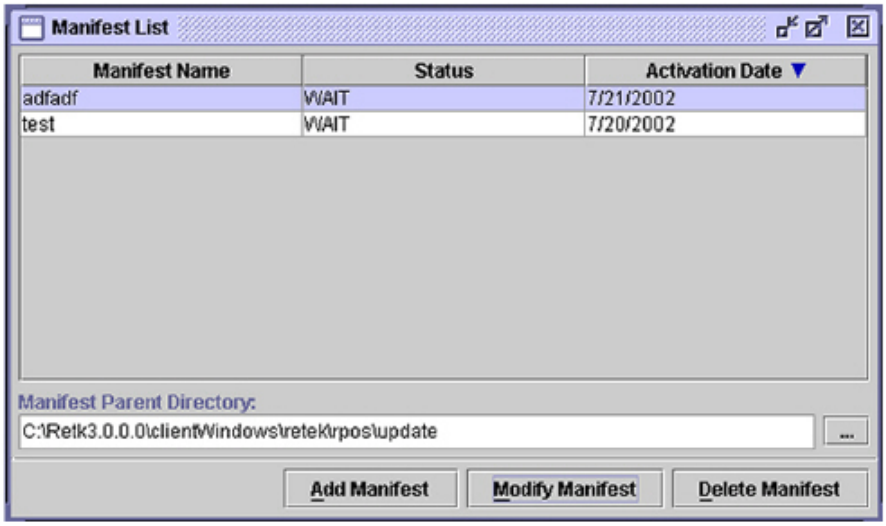
The following is the manifest process using RPOS Mission Control:

1. Create a manifest.
2. Add the manifest to the application servers.
3. Push the manifest to the clients.

Manifest list window

The **Manifest List** window is the working window for adding, modifying, and deleting manifests. To connect to the **Manifest List** window:

- **Select File>Manifest**. The **Manifest List** window is displayed.



The table below lists and describes the features and information displayed on the **Manifest List** window.

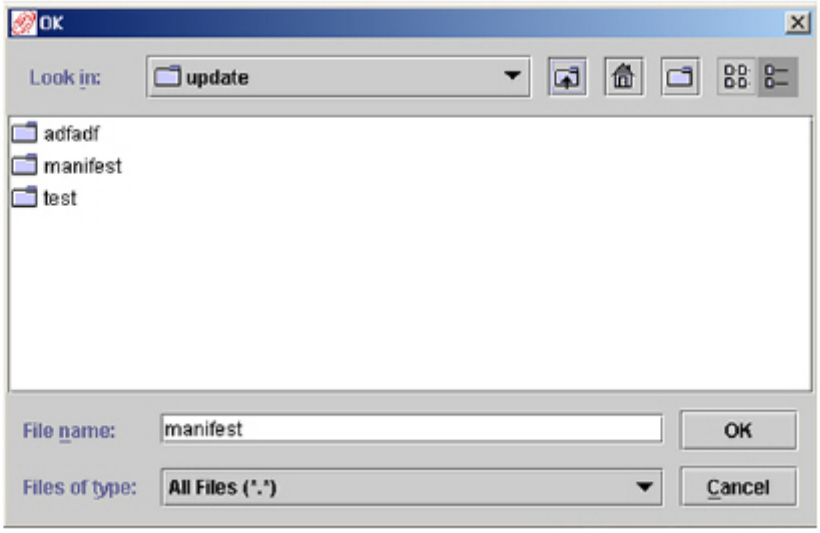
Manifest List Window Contents	
Feature	**Description**
Manifest Name	Displays the name of the manifest. This value is user-defined.
Status	Displays the status of the manifest. The manifest can have a status of remove, install, and delete. This value is user-defined.
Activation Date	Displays the activation date for the manifest. This value is user-defined.
Manifest Parent Directory	Displays the parent directory for the manifests.
Directory button	Allows you to open or modify the parent directory of the manifests
Add Manifest button	Allow you to create and add a new manifest.
Modify Manifest button	Allows you to modify an existing manifest.
Delete Manifest button	Allows you to delete a manifest.

Change the manifest parent directory

The manifest parent directory is the directory that stores the manifest files. To change the manifest parent directory:

1. In the **Manifest List** window, click the **Directory** button.

The **Directory** dialog box is displayed.



2. Select the parent directory for the new manifest.
3. Click **OK**. The **Manifest Parent Directory** field is updated with the parent directory for the new manifest.

Create a new manifest file

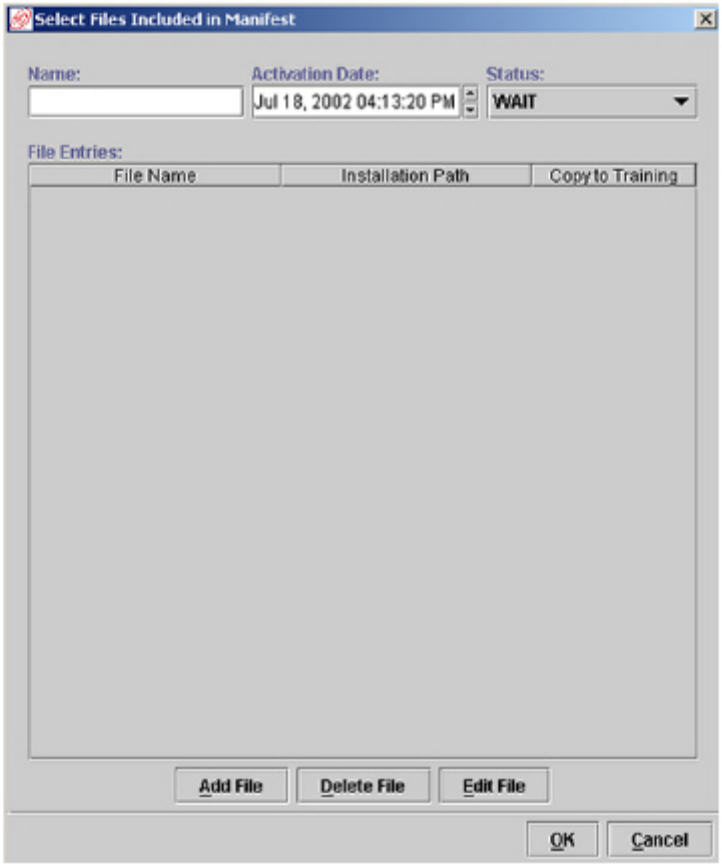
First, you have to create a new manifest in the **Manifest List** window. After you create the manifest, you must save the manifest on all the application servers by dragging and dropping the manifest from the **Manifest List** window onto the **Containers** window of the application servers.

A manifest contains software updates and data file updates that are pushed out to clients in an automated process. A manifest also contains an activation date and manifest status that provides clients with instructions on when to install the update files.

Creating a new manifest will not automatically update the application servers. The manifest must be saved on the application servers so the manifest files can be pushed out and installed on the clients.

To create a manifest file:

1. In the **Manifest List** window, click the **Add Manifest** button. The **Select Files Included in Manifest** dialog box is displayed.



2. In the **Name** field, type the manifest name.

Note: It is recommended that the name of the manifest be related to the information you are updating the clients with. This will help you better manage your manifest files.

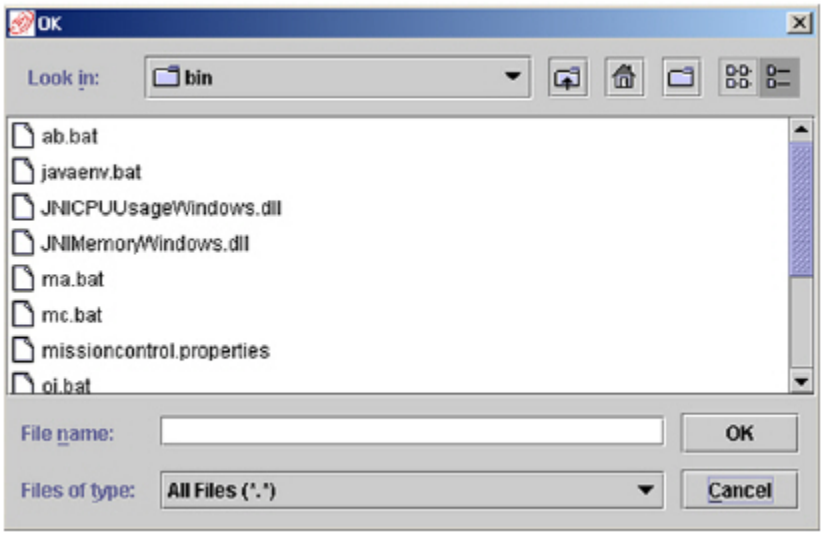
3. In the **Activation Date** field, enter the activation date for the manifest. The activation date is the date in which the clients will begin downloading the manifest.

Note: It is recommended that you set the activation date to a date prior to the actual installation to ensure all application servers are updated.

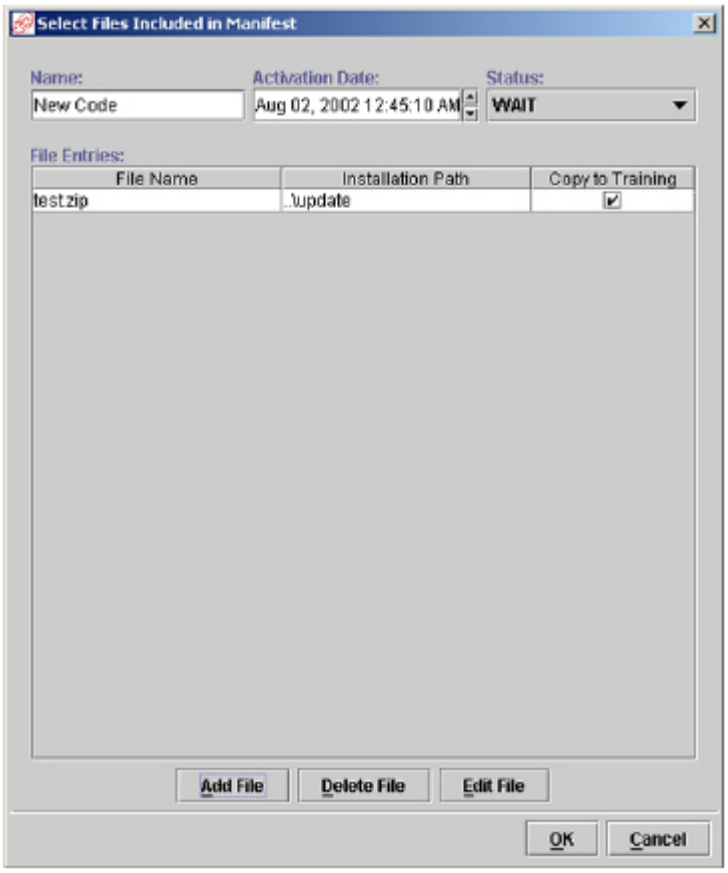
4. In the **Status** drop-down menu, select the appropriate status type for the manifest. By default, the **Wait** status is selected.

Note: When you create the manifest, it is recommended that you set the value for the manifest status to **WAIT**. Also, it is recommended that you do not change the status of the manifest until you are ready to update the application servers and push the manifest out to the clients.

5. Click the **Add File** button. The **Directory** dialog box is displayed.



6. Select the files you wish to include in the manifest.
7. Click **OK**. The selected files are displayed in the **Files Entries** table.



8. To add multiple files to a manifest, repeat steps 5-7.
9. On the **Selected Files Included in Manifest** dialog box, select **OK**. The manifest is displayed on the **Manifest List** window.

Save manifest on application servers

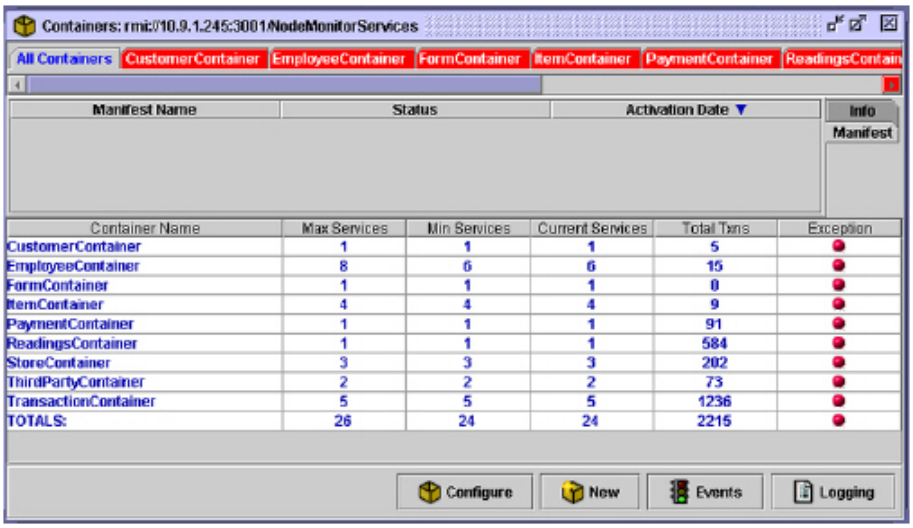
After you have created a manifest, you are ready to save the manifest on the application servers so it can be pushed out to the clients. To save the manifest on the application servers, you will have to drag and drop the manifest file from the **Manifest List** window onto the **Containers** window for each application server.

To save a manifest file on a application server:

1. Connect to all the application servers that require the manifest. The **Containers** window for each application server is displayed.

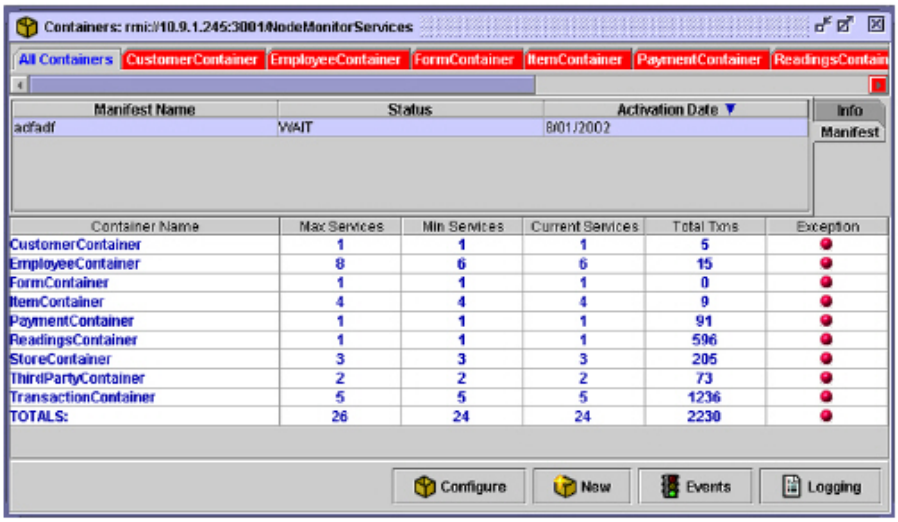
Note: All application servers must be updated with the manifest so each application server is running the most current and up-to-date version of the RPOS code and data files.

2. On the **Containers** window for each application server, click the **Manifest** tab. The **Manifest** table is displayed.



Note: You must click the **Manifest** tab on the **Containers** window before you can save a manifest on the application server.

3. In the **Manifest List** window, select the manifest you wish to update the application servers with. The selected manifest is highlighted in blue.
4. Drag and drop the manifest from the **Manifest List** window onto the **Containers** window of all the application servers. The manifest information appears in the **Manifest** table, and it is saved on the application servers.



5. To save another manifest onto the application servers, repeat steps 3-4.

Push a manifest to the clients

After the manifest has been saved on the application servers, you are ready to push the manifest to the clients by updating the application servers.

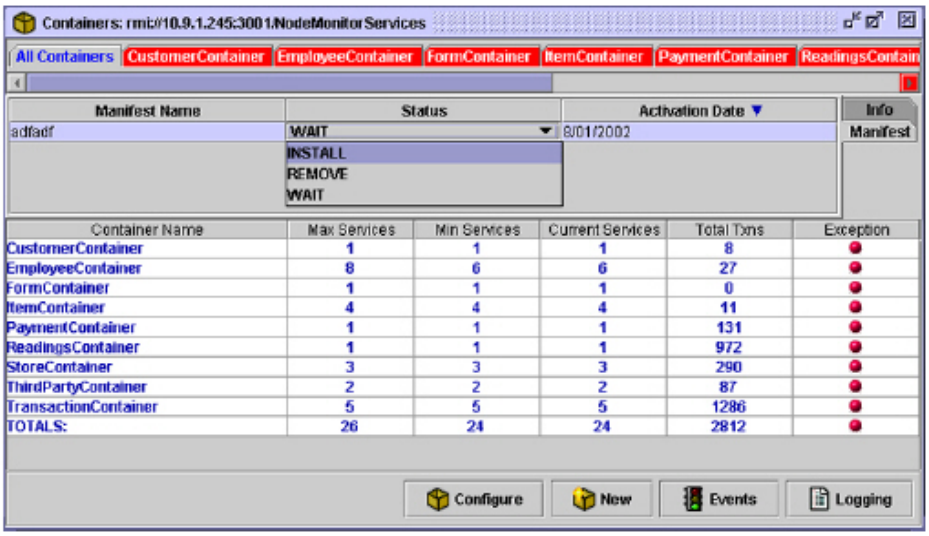
To install the manifest onto the clients:

1. Manually update the application servers with the manifest files.
2. Stop all the application servers.
3. Restart the application servers with the new version of RPOS.
4. Change the manifest status on all application servers to **INSTALL**. When the client checks the application servers for a new version of RPOS, it will see that the status as **INSTALL** and begin downloading and installing the new software files and data files.

To change the manifest status to INSTALL on the containers window

To change the manifest status to **INSTALL** on the **Containers** window:

1. In the **Containers** window on all the application servers, select the appropriate manifest. The selected manifest is highlighted in blue.
2. Click in the **Status** column. The **Status** drop-down menu is displayed.



1. In the **Status** drop-down menu, select **INSTALL**. The **Select Option** dialog box is displayed.
2. In the **Select Option** dialog box, select **OK**.

Modify a manifest

Mission Control allows you to modify a manifest from either the **Manifest List** window or the **Containers** window. If you make modifications to a manifest file in the **Manifest List** window after you have saved the manifest onto the application servers, you will have to replace the manifest that resides on the application servers with the updated manifest. After you have saved a manifest on an application server, you can still make modifications to it.

You can make the following changes to a manifest from the **Manifest List** window:

- Change the activation date of the manifest
- Change the manifest status
- Add files to a manifest
- Remove a file from a manifest
- Edit a file included in a manifest
- Copy the manifest to the training mode of RPOS

You can make the following changes to a manifest from the **Containers** window:

- Change the activation date of the manifest
- Change the manifest status

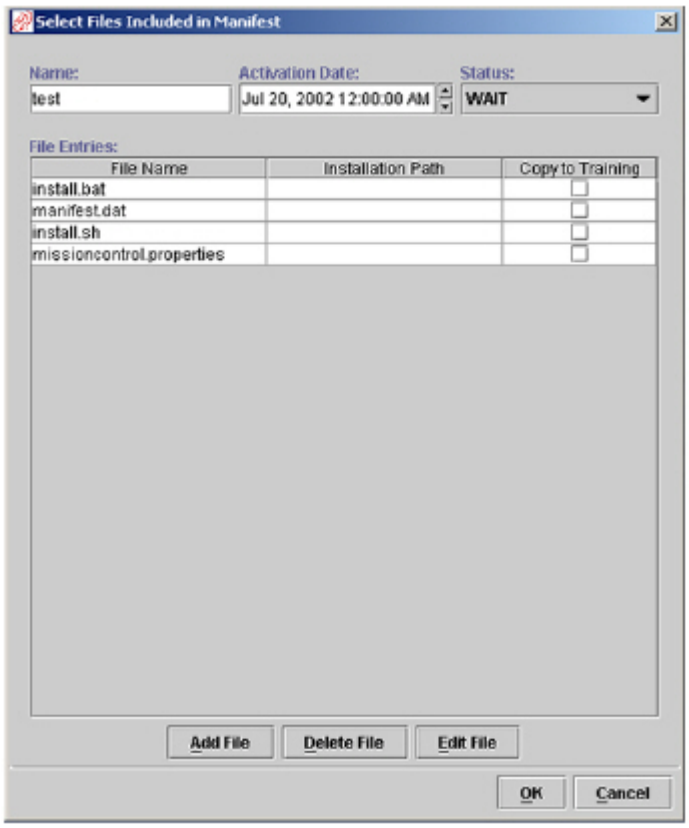
Change the manifest activation date

You can change the manifest activation date from either the **Manifest List** windows or the **Containers** window. If you change the activation date using the **Manifest List** window, it is recommended that you make these changes before you save the manifest onto an application server.

Change the manifest activation date from the manifest list window

To change the manifest activation date from the **Manifest List** window:

1. In the **Manifest List** window, select the manifest you wish to change the activation date of. The selected manifest is highlighted in blue.
2. Click the **Modify Manifest** button. The **Selected Files Included in Manifest** window is displayed.



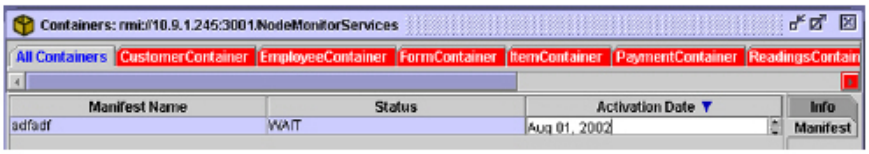
3. Click inside the **Activation Date** field.

4. To change the activation date, type the new activation information.
   OR
   Place the cursor in the desired area and click the up or down arrow. For example, if you want to change the month that the manifest will be activated, place the cursor inside the displayed month and click the up arrow to change the month.
5. Click **OK**. The new activation date is displayed in the **Manifest List** window.

Change the manifest activation date from the containers window

After saving the manifest onto the application servers, you can still change the activation date of the manifest. To change the manifest activation date from the **Containers** window:

1. In the **Containers** window of the application server you wish to change the manifest activation date of, select the appropriate manifest. The selected manifest is highlighted in blue.
2. Double click in the **Activation Date** column. The **Up** and **Down** arrows are displayed.



3. To change the activation date, type the new activation information.
   OR
   Place the cursor in the desired area and click the up or down arrow. For example, if you want to change the month that the manifest will be activated, place the cursor inside the displayed month and click the up arrow to change the month.

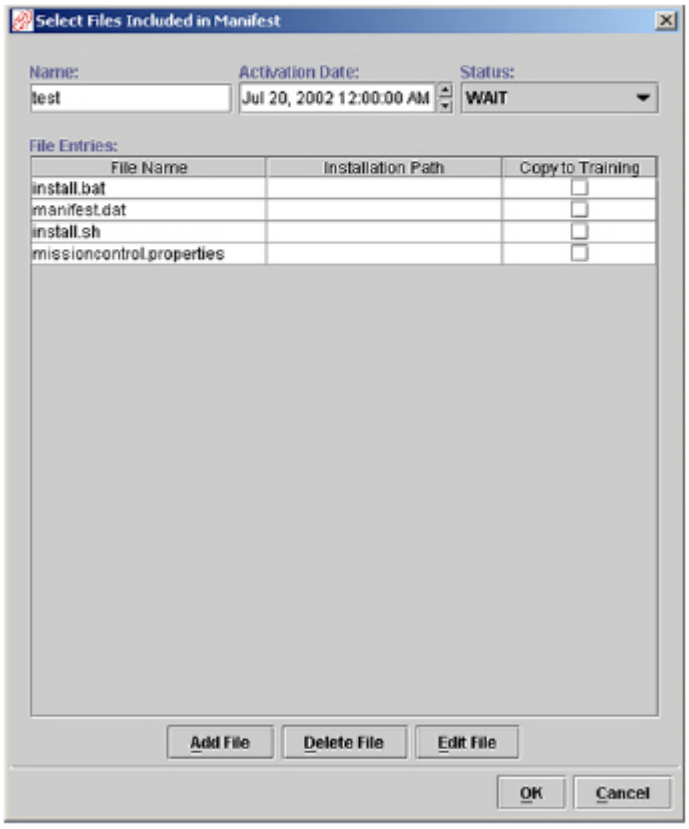
Change the manifest status

You can change the manifest status from either the **Manifest List** window or the **Containers** window. If you change the manifest status using the **Manifest List** window, it is recommended you make these changes before you save the manifest onto an application server.

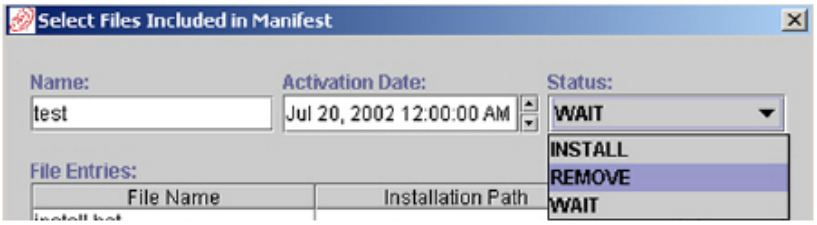
Note: It is recommended that you only change the manifest status on the **Containers** window of the application servers when you are ready to push the manifest to the clients.

To change the manifest status from the **Manifest List** window:

1. In the **Manifest List** window, select the manifest you wish to change the activation date of. The selected manifest is highlighted in blue.
2. Click the **Modify Manifest** button. The **Selected Files Included in Manifest** window is displayed.



3. In the **Status** drop-down menu, select the new manifest status.

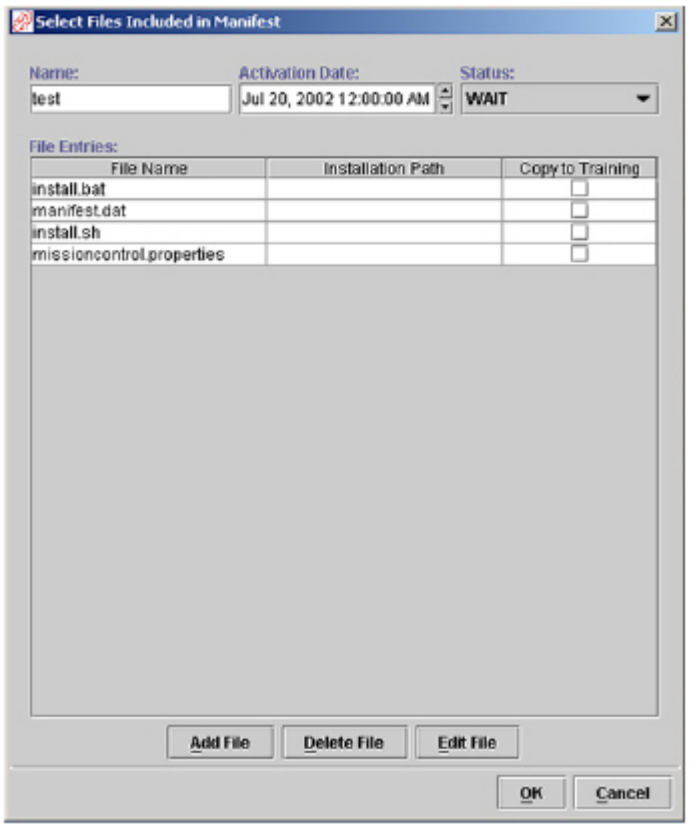


4. Click **OK**. The new manifest status is displayed in the **Manifest List** window.

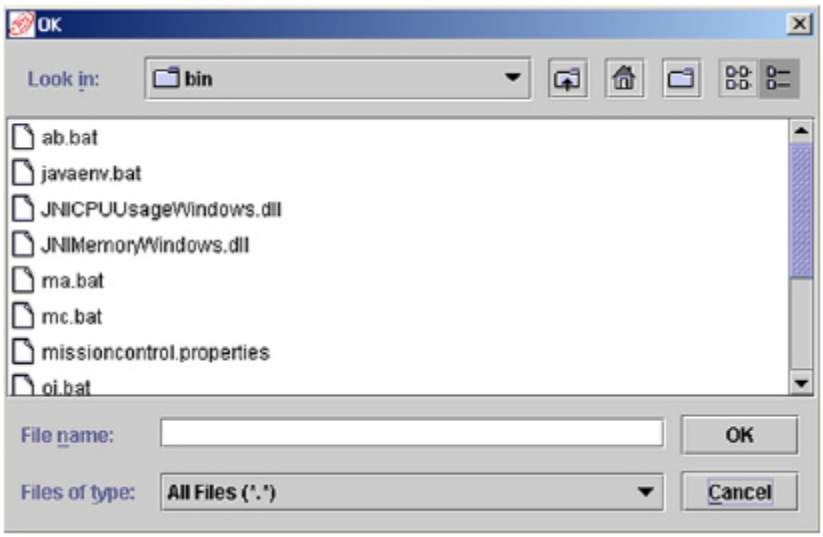
Add a file to a manifest

To add a file to a manifest:

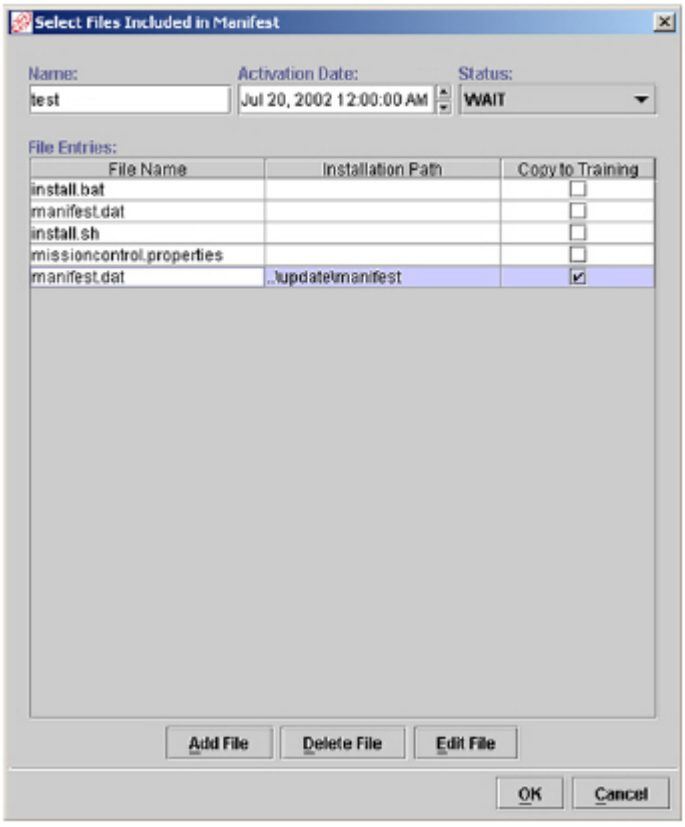
1. In the **Manifest List** window, select the manifest you wish to add a new file to. The selected manifest is highlighted in blue.
2. Click the **Modify Manifest** button. The **Selected Files Included in Manifest** window is displayed.



3. Click the **Add File** button. The **Directory** dialog box is displayed.



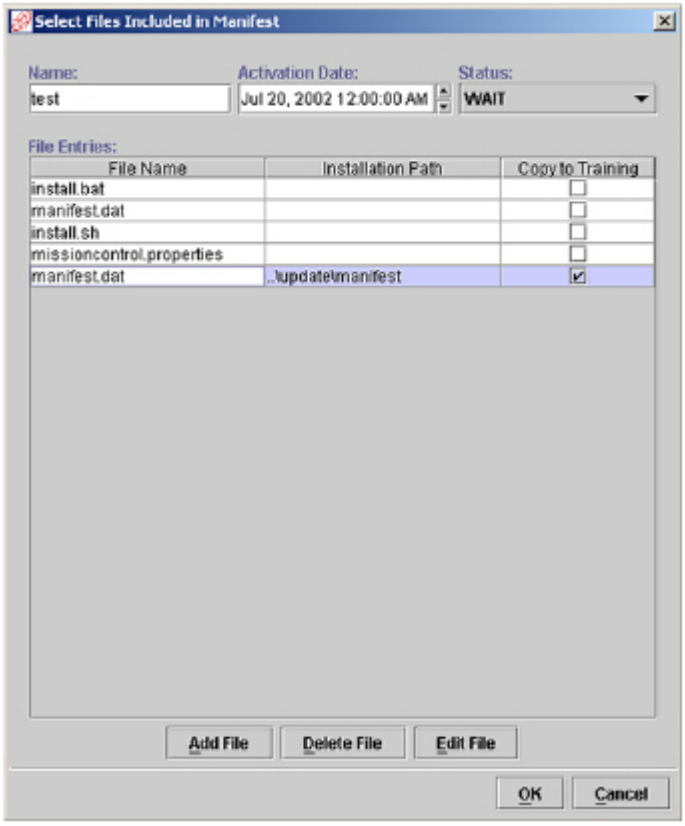
4. In the **Directory** dialog box, select the files you wish to add to the manifest.
5. Click **OK**. The selected files are displayed in the **Files Entries** table.



Delete a file from a manifest

To delete a file from a manifest:

1. In the **Manifest List** window, select the manifest that contains the file you wish to delete. The selected manifest is highlighted in blue.
2. Click the **Modify Manifest** button. The **Selected Files Included in Manifest** window is displayed.
3. In the **Selected Files Included in Manifest** window, select the file you wish to delete. The selected file is highlighted in blue.



4. Click the **Delete File** button. The **Delete File** dialog box is displayed.

Note: If you do not select a file before you click the **Delete** button, the **Select File to Delete** dialog box is displayed. To close the dialog box, select **OK**.

5. Click **OK**. The selected file is no longer displayed in the **Selected Files Included in Manifest** window.

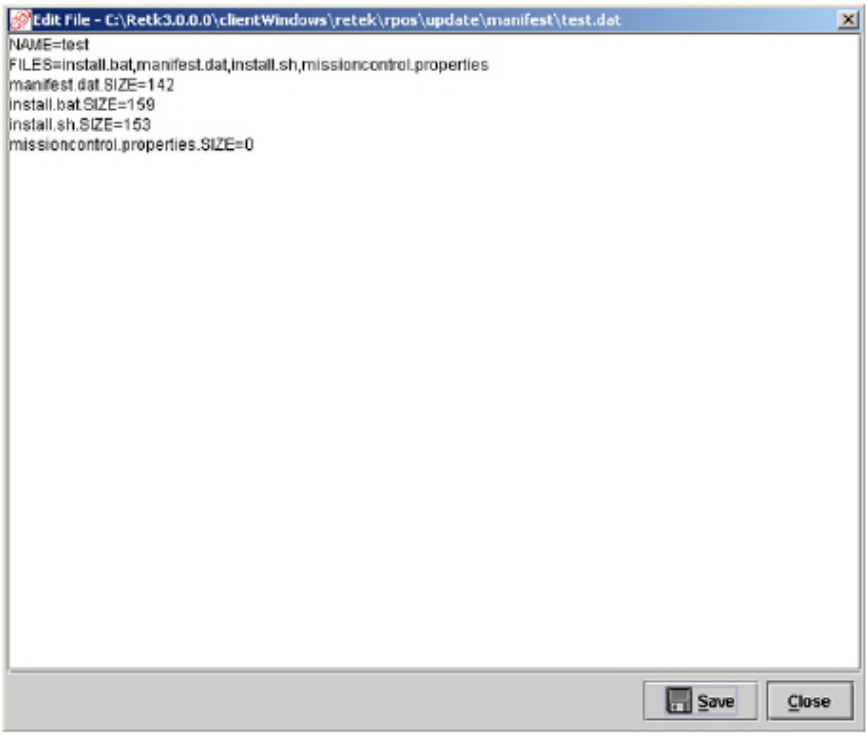
6. To delete more files from a manifest, repeat steps 3-5.
7. On the **Selected Files Included in Manifest** window, click **OK** to return to the **Manifest List** window.

Edit a file included in a manifest

Mission Control allows you to open and edit a file that you are including in a manifest. To edit a file included in a manifest:

1. In the **Manifest List** window, select the manifest that contains the file you wish to edit. The selected manifest is highlighted in blue.
2. Click the **Modify Manifest** button. The **Selected Files Included in Manifest** window is displayed.
3. Select the file you wish to edit. The selected file is highlighted in blue.

4. Click the **Edit File** button. The **Edit File** dialog box is displayed and the file is displayed in a text editor.



5. Modify or edit the file.
6. After you have made all your changes, click the **Save** button. The **Save File** dialog box is displayed.
7. Click the **Save** button. The **File Already Exists** dialog box is displayed.
8. To replace the existing file with the updated file, click **OK**.
9. To edit more files from a manifest, repeat steps 3-8.
10. On the **Selected Files Included in Manifest** window, click **OK** to return to the **Manifest List** window.