CA Nimsoft® Unified Management™ Portal

User Documentation
7.5
Legal Notices

This online help system (the "System") is for your informational purposes only and is subject to change or withdrawal by CA at any time.

This System may not be copied, transferred, reproduced, disclosed, modified or duplicated, in whole or in part, without the prior written consent of CA. This System is confidential and proprietary information of CA and protected by the copyright laws of the United States and international treaties. This System may not be disclosed by you or used for any purpose other than as may be permitted in a separate agreement between you and CA governing your use of the CA software to which the System relates (the “CA Software”). Such agreement is not modified in any way by the terms of this notice.

Notwithstanding the foregoing, if you are a licensed user of the CA Software you may make one copy of the System for internal use by you and your employees, provided that all CA copyright notices and legends are affixed to the reproduced copy.

The right to make a copy of the System is limited to the period during which the license for the CA Software remains in full force and effect. Should the license terminate for any reason, it shall be your responsibility to certify in writing to CA that all copies and partial copies of the System have been destroyed.

TO THE EXTENT PERMITTED BY APPLICABLE LAW, CA PROVIDES THIS SYSTEM “AS IS” WITHOUT WARRANTY OF ANY KIND, INCLUDING WITHOUT LIMITATION, ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE OR NONINFRINGEMENT. IN NO EVENT WILL CA BE LIABLE TO YOU OR ANY THIRD PARTY FOR ANY LOSS OR DAMAGE, DIRECT OR INDIRECT, FROM THE USE OF THIS SYSTEM, INCLUDING WITHOUT LIMITATION, LOST PROFITS, LOST INVESTMENT, BUSINESS INTERRUPTION, GOODWILL, OR LOST DATA, EVEN IF CA IS EXPRESSLY ADVISED IN ADVANCE OF THE POSSIBILITY OF SUCH LOSS OR DAMAGE.

The manufacturer of this System is CA.

Provided with “Restricted Rights.” Use, duplication or disclosure by the United States Government is subject to the restrictions set forth in FAR Sections 12.212, 52.227-14, and 52.227-19(c)(1) - (2) and DFARS Section 252.227-7014(b)(3), as applicable, or their successors.

Copyright © 2014 CA. All rights reserved. All trademarks, trade names, service marks, and logos referenced herein belong to their respective companies.

Legal information on third-party and public domain software used in the Nimsoft Monitor solution is documented in Nimsoft Monitor Third-Party Licenses and Terms of Use (http://docs.nimsoft.com/prodhelp/en_US/Library/Legal.html).
Contact CA Support

For your convenience, CA Technologies provides one site where you can access the information that you need for your Home Office, Small Business, and Enterprise CA Technologies products. At http://ca.com/support, you can access the following resources:

- Online and telephone contact information for technical assistance and customer services
- Information about user communities and forums
- Product and documentation downloads
- CA Support policies and guidelines
- Other helpful resources appropriate for your product

Providing Feedback About Product Documentation

Send comments or questions about CA Technologies Nimsoft product documentation to nimsoft.techpubs@ca.com.

To provide feedback about general CA Technologies product documentation, complete our short customer survey which is available on the CA Support website at http://ca.com/docs.
Contents

Chapter 1: Welcome 17
Getting Started with UMP .................................................................17
   The Portal .......................................................................................17
   Create a Page .................................................................................20
   Managing Pages ...........................................................................21
   Error Console ..............................................................................21
Welcome to Unified Management Portal Help ....................................22
   How to Navigate the Help Topics ................................................22
   How to Print Multiple Help Topics ..............................................23

Chapter 2: Account Admin 25
Types of Users ....................................................................................25
Account Admin Pane ...........................................................................26
New Account/Edit Account Dialog ....................................................27
Edit ACLs Dialog .............................................................................27
   Edit Alarm Filter Dialog ............................................................28
   UMP Views ................................................................................29

Chapter 3: Change Password 31

Chapter 4: Admin Console Guide 33

Chapter 5: Alarm Console 35
Alarm Console Toolbars ......................................................................35
   General Bar ....................................................................................35
   Command Bar ................................................................................36
   Alarm Filter Bar ............................................................................37
   History Bar ...................................................................................38
Alarm Console Table ..........................................................................40
Popup Menus ....................................................................................42
   Toolbar Popup Menu .....................................................................42
   Alarm Console Table Popup Menu .............................................43
Dialogs .............................................................................................45
   Actions Dialog .............................................................................45
   Adobe Flash Player Settings Dialog .........................................46
Chapter 6: CA APM Cloud Monitor

Importing the LAR File for CA APM Cloud Monitor ................................................................. 57
Cloud Monitor Server Location Configuration ........................................................................... 58
Configuring Access to the CA APM Cloud Monitor Portlet .................................................... 59

Chapter 7: Dashboard

Create a Dashboard: Overview ................................................................................................. 62
View Modes ............................................................................................................................. 63
Prepare the Canvas .................................................................................................................. 63
  Size the Canvas ................................................................................................................... 63
  Set the Background Properties ......................................................................................... 64
  Set a Background Image ................................................................................................. 64
Set the Grid Functions ......................................................................................................... 65
Add a Widget ......................................................................................................................... 65
Chapter 8: Custom Dashboards (Legacy) 107

Chapter 9: Dashboard Designer (Legacy) 111

Chapter 10: Dynamic Views 183
Chapter 16: Nimsoft Remote Admin

Migration for Legacy Users ...................................................................................................................................... 228
Troubleshooting Nimsoft Remote Admin ................................................................................................................ 228
USM Templates Do Not Deploy ........................................................................................................................... 229

Chapter 17: Performance Reports Designer

Understanding Performance Data ........................................................................................................................... 231
Viewing Raw Performance Data ............................................................................................................................ 232
QoS Selection Tabs ................................................................................................................................................... 233
Filters Tab ................................................................................................................................................................ . 234
User Tags .................................................................................................................................................................. 235
Charts Pane .............................................................................................................................................................. 236
Automatic Replacement of Fields in PRD Templates ............................................................................................... 241
Toolbar ..................................................................................................................................................................... 242
PDF Preferences Dialog ..................................................................................................................................... 244
Save Dialog ........................................................................................................................................................ 244
Preferences Dialog ............................................................................................................................................ 246
Define Reports with a URL ....................................................................................................................................... 246
Troubleshooting PRD ................................................................................................................................................ 251
Performance Reports May Convert Incorrectly to PDF ............................................................................................. 252

Chapter 18: Relationship Viewer

Overview .................................................................................................................................................................. 253
About the Relationship Viewer ........................................................................................................................... 253
Accessing the Relationship Viewer ....................................................................................................................... 254
Using the Relationship Viewer ............................................................................................................................ 255
Changing What You See .................................................................................................................................... 255
Relationship Viewer Control Bar .......................................................................................................................... 256
Viewing Object Details ...................................................................................................................................... 264
Launching the Relationship Viewer using the WebContent Portlet ......................................................................... 265

Chapter 19: Reports

Reports Overview ..................................................................................................................................................... 269
Quality of Service Reports ........................................................................................................................................ 269
Service Level Agreement Reports ............................................................................................................................ 272
The Unavailability Report ...................................................................................................................................... 272
Chapter 23: SLM

SLM Overview .......................................................................................................................................................... 300
  Data Collection and Compliance Calculation ................................................................................................. 301
  Compliance Percentage ................................................................................................................................. 301
  Multi-Series Calculation Profile ................................................................................................................... 305
  Calculation Examples .................................................................................................................................... 306
Using SLM .......................................................................................................................................................... 311
  Setting Up a Probe to Deliver Quality of Service Data .................................................................................. 312
  Overview of Creating Service Level Agreements ......................................................................................... 312
  Example - Defining a Simple Service Level Agreement ............................................................................... 345
  Creating an SLA Using the SLA Wizard .......................................................................................................... 348
  Creating a QoS Monitoring Profile .............................................................................................................. 355
  Baseline Value Definition ............................................................................................................................. 355
  Viewing the Quality of Service (QoS) Data .................................................................................................... 361
  Exporting the QoS Data Series ..................................................................................................................... 363
  Exporting QoS Data Using Export QoS Data Wizard .................................................................................. 364
  Data Management ....................................................................................................................................... 364
  Watching SLA Calculations .......................................................................................................................... 372
SLM Database Structure .................................................................................................................................. 374
  QoS Data Tables .......................................................................................................................................... 374
  SLA Configuration Tables ............................................................................................................................ 375
  SLA Calculation Tables ............................................................................................................................... 375
  Account Tables .......................................................................................................................................... 376
  Sending SQL Queries to the Database .......................................................................................................... 377
SLM Interface Reference .................................................................................................................................. 379
  The Menu Bar ............................................................................................................................................. 380
  Preferences Dialog .................................................................................................................................... 382
  The Navigation Pane .................................................................................................................................. 383
  Main Window Pane .................................................................................................................................... 389
SLM Terms and Definitions ............................................................................................................................... 390
  Automatic (Interval) data type ...................................................................................................................... 390
  Asynchronous data type .............................................................................................................................. 390
  Calculation method ................................................................................................................................... 390
  Compliance period ..................................................................................................................................... 390
  Compliance Percentage ............................................................................................................................... 391
Chapter 24: Unified Reporter .................................................................................................................. 397

Chapter 25: Unified Service Manager .................................................................................................... 399

Access to USM ........................................................................................................................................... 402
Navigating in USM ...................................................................................................................................... 402
Views ........................................................................................................................................................... 402
Search for Computer Systems ...................................................................................................................... 407
Pop Out Graphs ........................................................................................................................................... 409
Launch Relationship Viewer ......................................................................................................................... 409
Geo View .................................................................................................................................................... 410
Creating a Geo View ................................................................................................................................... 411
Discovery .................................................................................................................................................... 412
Running the Discovery Wizard ..................................................................................................................... 412
Run File-based Import ................................................................................................................................ 421
View Discovered Systems ............................................................................................................................. 428
More Information about Discovery ................................................................................................................. 429
Automatic Robot Deployment ....................................................................................................................... 429
Deploy Robots Automatically ...................................................................................................................... 430
Deploy Robots to Systems Listed in a File .................................................................................................... 431
View the Status of Robot Deployment ......................................................................................................... 442
Getting Started with Service-Oriented Configuration .................................................................................. 443
Set Up Service-Oriented Configuration ....................................................................................................... 443
SOC Probes Configuration Reference ............................................................................................................ 444
Working with Groups ................................................................................................................................ 445

Data Types .................................................................................................................................................. 392
Hub .............................................................................................................................................................. 392
NULL value ................................................................................................................................................ 392
Operating period ........................................................................................................................................ 392
Probes ....................................................................................................................................................... 393
Robot ......................................................................................................................................................... 393
Service Level Agreement (SLA) .................................................................................................................. 393
SLO ............................................................................................................................................................ 393
SLM ............................................................................................................................................................. 393
QoS ............................................................................................................................................................. 394
QoS object ................................................................................................................................................ 394
QoS constraint .......................................................................................................................................... 394
Troubleshooting SLM ................................................................................................................................. 394
SLM May Allow Invalid Logical Expressions ............................................................................................... 394
The SLM Portlet Does Not Record a QoS Object ....................................................................................... 395
SLM Portlet Displays Incorrect Data Set .................................................................................................... 395

Contents 13
Chapter 29: Troubleshooting

Single Time Zone Required ................................................................. 509
LDAP Authentication with Active Directory Groups ............................. 510
Custom Logo Missing ........................................................................ 510
Unified Service Manager Displays Non-localized Text ....................... 510
Relationship Services Not Supported on Solaris ................................. 511
I Cannot Access a Wiki Page I Created ............................................. 511
Page Titles May Contain Garbled Text .............................................. 512
Garbled Text in Console Installer ....................................................... 512
Relationship Viewer Dependent upon Root Cause Analysis and Topology Manager ................................................. 513
Close Browser When Logging Out ....................................................... 513
Size of Dashboard Panels ................................................................. 513
Do Not Edit QoS Properties in SLM Portlet ..................................... 513
Turn Off Compatibility View with IE 8 or 9 ..................................... 514
Internet Explorer 8 Performance Issue .............................................. 514
Internet Explorer 8 Displays Page Title Incorrectly ............................ 514
Portlet Not Resizing in Chrome ......................................................... 515
User Names ...................................................................................... 515
Internet Explorer 8 Cross-Site Scripting Issue .................................. 516
Login Fails on Solaris ....................................................................... 517
Slow Performance on Internet Explorer 7/8 ..................................... 517
Non-ASCII Characters Stored Incorrectly in the Database ............... 517
Numeric Login IDs Do Not Work ................................................----- 518
I Cannot Access the Control Panel .................................................. 518
Logging In Takes Me to the Login Page .......................................... 519
CSV File is Incorrect in Chinese or Japanese .................................. 519

Index .................................................................................................. 521
Chapter 1: Welcome

This section contains information about getting started with the Unified Management Portal (UMP) and about using the UMP online help.

For the complete set of documentation for CA Nimsoft products, see the Nimsoft Library.

This section contains the following topics:

- Getting Started with UMP (see page 17)
- Welcome to Unified Management Portal Help (see page 22)

Getting Started with UMP

This topic tells you how to start displaying and viewing data in the Unified Management Portal (UMP). You can view the default pages that come with UMP, or you can create and save pages with a custom layout.

The Portal

The CA Nimsoft Unified Management Portal (UMP) is where you:

- Open and manage Service Desk tickets
- Monitor and manage computer systems
- Graph QoS data
- View and manage alarms
- Create SLAs and view SLA performance reports
- Create, view, and schedule reports
- Create and view custom dashboards
- Manage users
The following list describes the portlets available. Some portlets require an additional purchase.

- **Account Admin** - Allows you to manage users and user privileges, including assigning access control lists (ACLs) to contacts within an account.

  **Note**: The ACL must be marked as available to an account in order to assign the ACL to contacts within that account. For account contact users, the alarms listed are filtered by origin – users from an account see only alarms from an origin which the account has ownership of. Discovery information is handled in a similar way: a contact user sees only nodes from an origin which the account has ownership of.

- **Admin Console** - Allows you to configure probes and manage your CA Nimsoft Monitor infrastructure (hubs, robots, and probes).

- **Alarm Console** - Allows full viewing, filtering, and managing of alarms.

- **CA APM Cloud Monitor** - Allows you to monitor web sites and cloud services from around the globe. Measures the status of your transactions and services from more than 60 locations.

- **Custom Dashboards** - Allows you to:
  - Access your custom dashboards, which display the QoS data and alarms that are important to you in managing your network.
  - View your alarms.
  - View the Dynamic Views, which display the state (such as alarm level, performance, etc.) of the monitored computer systems on your network.

- **Dashboard Designer** - Allows you to design custom dashboards to display selected QoS data and alarms from some or all of your monitored computer systems.

- **Dynamic Views** - Displays automatically generated dashboards for the computer systems discovered on your network. The dashboards display QoS information such as memory and CPU utilization.

- **Flow Analysis** - Displays network traffic flow data (NetFlow data measured and collected on router interfaces) organized into TopN reports for router interfaces, hosts, and applications.

- **List Designer** - Allows you to design lists to be displayed in the List Viewer application.

- **List Viewer** - Displays data in a table format. The data can be in the form of text, numbers, gauges, alarms, or graphs. Two kinds of lists can be created, detail or group. In detail lists, each row displays data for a single host or target. In group lists, each row displays aggregated data for a group of systems. When creating group lists, you can specify a drilldown list for each group. Typically, the drilldown list contains the data for the individual hosts or targets that make up the group.
■ My Tickets - Allows you to view basic information about your CA Nimsoft Service Desk tickets. Unlike the Service Desk, My Tickets can be displayed on the same page as other portlets, allowing you to view your tickets alongside other key data for your organization.

■ CA Nimsoft Remote Admin - Formerly used for discovery, this portlet's functionality is replaced by the Discovery features in the Unified Service Manager portlet. However, this portlet remains available for legacy customers.

■ Performance Reports Designer - Allows you to see a visual representation of QoS data. You select the host, QoS measurement, target, and time range, and the data is displayed as a graph. You can display multiple measurements on a single graph, and can view multiple graphs at a time. You can choose the graph format (line, area, or column graph) and can maximize the graph to view it at a larger size. You can save a set of graphs as a report to view later.

■ Relationship Viewer - Provides a facility for applications to display the status of and relationships between objects in various ways. For example, the Root Cause Analysis and Topology Manager application, if installed, uses the Relationship Viewer to display symbols that represent network devices, show the connections between those devices, and indicate the status of each device and connection.

■ Reports - Contains Quality of Service (QoS) reports and Service Level Agreement (SLA) reports. QoS reports must be manually created in the Performance Reports Designer portlet. SLA reports are created automatically after you create SLAs in the Service Level Manager (SLM) portlet.

■ Report Scheduler - Allows you to schedule reports to run at specified times. The reports are created in PDF format and can be sent via email or FTP, or stored on a server.

■ Service Desk - Provides action-based workflows based on ITIL standards that allow you to coordinate incident response and proactive IT management. Integrates data from CA Nimsoft Monitor to allow you to view data from operations and service management together.

■ SLA Reports - Provides SLA reports to show overall compliance, SLA details, SLOs, SLA compliance trend, and graphs on the SLO data.

■ SLM - Provides a set of tools to monitor and ensure the validity of a set of SLAs for internal and external customers. It is made up of Service Level Agreements (SLAs), Service Level Objectives (SLOs), and Quality of Service (QoS) data.

■ Unified Reports - Provides advanced reporting. You can run standard reports that are provided, or you can create your own reports in the Report Designer. This is an optional component that you must purchase.

■ Unified Service Manager - Allows you to set up monitoring for groups of computer systems and to view the data collected. You can create groups of computer systems and then assign monitoring templates to them. You can also view alarms and perform discovery of devices on your network.
Create a Page

You can create custom pages where you control which portlets are displayed and the layout of the page. Two privacy settings are available:

**Private**

Pages display when you first log in. Only you can see private pages.

**Public**

Pages are accessed by choosing Go to, My Public Pages from the menu bar. Anyone on the Internet can see public pages.

To create a page:

1. [Add a page](#) (see page 20).
2. [Select a layout](#) (see page 20).
3. [Add portlets](#) (see page 20).

Add a Page

1. Choose Add, Page from the menu bar.
2. Enter a name for the page in the text box that is displayed next to the page tabs.
3. Click the green checkmark next to the text box.
4. Click the tab for the page.
   The new page appears.

Select a Layout

1. Choose Manage, Page Layout from the menu bar.
2. Select the layout that you want.
3. Click Save.

Add Portlets

1. Choose Add, Portlet from the menu bar.
2. Click the + sign next to Monitoring.
3. Drag a portlet to the position on the page where you want to display it, or click Add next to the portlet.

To add more portlets to your layout, repeat these steps.
Managing Pages

You can change the settings for your pages, including permissions, by choosing Manage, Control Panel from the menu bar.

Error Console

The Unified Management Portal has an error console where you can view error messages. If errors have been generated, an icon is present on the application toolbar. The icon represents the highest severity level of messages generated:

- = Informational warning
- = Error message

Click the icon to display the error console:

Double-click a message to view detailed information about the error:
Welcome to Unified Management Portal Help

This online Help system contains the how-to and reference information you need to use the Unified Management Portal (UMP) successfully. This first Help topic explains how to navigate and print the topics in this Help system.

If you are already comfortable using the online Help, you can either use the Contents, Index, or Search features in the left frame to find the Help you need now.

How to Navigate the Help Topics

The online Help is an HTML-based system that runs in standard Web browsers.

This Help system provides the following navigation features in the left frame of your browser window to help you quickly find information:

- Contents (see page 22)
- Index (see page 23)
- Search (see page 23)

The following subsections describe how to use each of these navigation features.

Using the Contents Feature

The Contents feature lists all the books and topics in this Help system in an expandable/collapsible tree hierarchy. This feature lets you view and navigate through the information in this Help system.

By default, the Contents list is displayed in the left frame of your browser window when the Help system first opens.

If the Contents list is not currently displayed in the left frame of your browser window, click the Contents tab at the top of the left frame.
Using the Index Feature

The Index feature lists primary and secondary keywords (UMP terms and common synonyms) used in this Help system. This feature gives you a quick way to find information on specific topics within the Help system.

To use the Index feature

Click the Index tab at the top of the left frame. Then, scroll to and click the keyword you want, or click on a letter at the top to jump to words beginning with that letter.

Using the Search Feature

The Search feature lets you find all topics containing a particular word or words within this Help system. This feature uses a full-text search engine that is built into the Help system and runs locally on your system.

To use the Search feature

Click the Search tab at the top of the left frame. Then, in the text box, type the word or words you want to search for and click the Search button. When the search completes, scroll to and click the Help topic you want.

How to Print Multiple Help Topics

To enable you to print multiple topics from this Help system, the content is provided in an Adobe® Acrobat® (PDF formatted) file. This file is available from the help title page.
Chapter 2: Account Admin

The Account Admin portlet allows administrators to manage account contact users and access control lists (ACLs). You must have the ACL permission *Account Administration* to access the Account Admin portlet.

In Account Admin, you can add, modify, or delete accounts and account contacts (users), and set passwords for account contacts. You can also add, modify, or delete ACLs, and set the permissions for the ACLs.

Changes that you make to ACLs in the Account Admin portlet are reflected in Infrastructure Manager, as both interfaces write to the same database.

This section contains the following topics:

- **Types of Users** (see page 25)
- **Account Admin Pane** (see page 26)
- **New Account/Edit Account Dialog** (see page 27)
- **Edit ACLs Dialog** (see page 27)
- **Change Password** (see page 31)

Types of Users

Two types of users exist in the CA Nimsoft Monitor solution—*regular Nimsoft* users and *account contact* users. These two types of users allow Nimsoft Monitor to provide access control that supports data segregation and multi-tenancy.

Regular Nimsoft users can manage other regular Nimsoft users, accounts, and account contact users. In addition, regular Nimsoft users can access Infrastructure Manager or Admin Console to manage the Nimsoft Monitor infrastructure. In a multi-tenant environment, managed service providers (MSPs) can create regular Nimsoft users who have administrative privileges to manage customer accounts and account contact users.

Account contact users are managed in the Account Admin portlet. Account contact users, even those with elevated privileges, only have access to their own account, and cannot manage the Nimsoft Monitor infrastructure. MSPs with multiple customers can create an account for each, and one or more account contact users with permissions to administer the account.
## Account Admin Pane

The following fields and icons in the Account Admin pane allow you to manage accounts and account contact users.

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Account pull-down</td>
<td>Select a user account. The users for that account are displayed in the table.</td>
</tr>
<tr>
<td>Add account icon</td>
<td>Click to open the <a href="#">New Account Dialog</a> (see page 27), where you can create user accounts.</td>
</tr>
<tr>
<td>Edit account icon</td>
<td>Click to open the <a href="#">Edit Account Dialog</a> (see page 27), where you can edit user accounts.</td>
</tr>
<tr>
<td>Delete account icon</td>
<td>Click to delete a user account.</td>
</tr>
<tr>
<td>Help icon</td>
<td>Click to view the online help for the Account Administration application.</td>
</tr>
<tr>
<td>Action icons</td>
<td>= Edit user. The fields in the table become editable. Make your changes and then click the (Save) icon. Or, click the (Cancel) icon to exit without saving your changes.</td>
</tr>
<tr>
<td></td>
<td>= Set Password. Opens the Change password dialog. Passwords must be at least 6 characters long.</td>
</tr>
<tr>
<td></td>
<td>= Delete user. Removes the user.</td>
</tr>
<tr>
<td>Login ID column</td>
<td>Login ID for the user.</td>
</tr>
<tr>
<td>First Name column</td>
<td>First name of the user.</td>
</tr>
<tr>
<td>Last Name column</td>
<td>Last name of the user.</td>
</tr>
<tr>
<td>Email column</td>
<td>Email address of the user.</td>
</tr>
<tr>
<td>ACL column</td>
<td>Access Control List (ACL) assigned to the account contact. ACL permissions determine which applications a user can view, which actions a user can take with alarms, and which administrative permissions, if any, the user has. For information on setting permissions for applications, see <a href="#">Setting Permissions for UMP</a> (see page 504).</td>
</tr>
</tbody>
</table>

Click the (Edit ACLs) icon in the column header to display the [Edit ACLs Dialog](#) (see page 27).
Add user icon

Click to add an account contact user. Enter user information in the new row at the bottom of the table, and select an ACL from the drop-down menu. When you click the (Save) icon, you are prompted to enter a password for the account contact user.

**Note:** Login IDs can only contain numeric characters and lowercase alphabetic characters. In addition, login IDs must begin with a lowercase alphabetic character. Special characters cannot be used. Passwords must be at least 6 characters long.

---

**New Account/Edit Account Dialog**

The **New Account** and **Edit Account** dialogs allow you to create or edit user accounts.

Enter the name and address information for the user. The **Ownership** is the origin assigned to the account, which determines what information is visible for account contact users. You can assign more than one origin to an account.

If you are an MSP, for example, you might designate the primary hub for each customer as the origin, thereby separating customer information. In USM, account contact users can only see alarms and discovered systems from origins assigned to the account.

**Edit ACLs Dialog**

The **Edit ACLs** dialog allows you to create, modify, or delete ACLs. You can choose which permissions to assign to each ACL. To access the dialog, click the **Edit ACLs** icon in the ACL column header of the Account Admin table.

For a list of permissions required to use UMP applications, see [Setting Permissions for UMP Portlets](#) (see page 504).

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACL list</td>
<td>Lists the ACLs that have been defined, using either the Account Admin application or Infrastructure Manager.</td>
</tr>
<tr>
<td>Check boxes</td>
<td>Click to select the permissions to be assigned to the ACL. Click the check box in the column header to toggle between selecting all and de-selecting all permissions.</td>
</tr>
<tr>
<td>Permission column</td>
<td>Name of the permission.</td>
</tr>
<tr>
<td>Description column</td>
<td>Description of the permission.</td>
</tr>
</tbody>
</table>

---

Chapter 2: Account Admin 27
### Type column
Type of permission.

### Access column
Type of access granted for that permission:
- **read** - Can view data only
- **write** - Can view and modify data
- **admin** - Can view and modify data, and make administrative changes
- **super** - Full access

---

### Add ACL icon
Click to create a new ACL, then enter a name for the ACL. Once it is created, click the check boxes to assign permissions to the ACL.

---

### Copy ACL icon
Click to copy an existing ACL, then enter a name for the ACL. Once it is created, you can click the check boxes to change the permissions for the ACL.

---

### Rename ACL icon
Click to change the name of the ACL.

---

### Delete ACL icon
Click to delete the selected ACL.

---

### LDAP group button
Click to select an LDAP group of users who will be authenticated as NMS users with the ACL currently selected.

---

### Account link button
Click to select an account to associate with an LDAP group. If an account is selected, the LDAP group will be contact users in the selected account. If no account is selected, the LDAP group will not be account contact users.

---

### Alarm Filter button
Click to launch the *Edit Alarm Filter Dialog* (see page 28), and set alarm filter criteria and operators for the ACL currently selected.

---

### UMP Views button
Click to launch the Configure UMP Views dialog, and configure which tabs and views are available in USM for the ACL currently selected.

---

### OK button
Click to save changes and exit the dialog.

---

### Cancel button
Click to exit the dialog without saving changes.

### Edit Alarm Filter Dialog

You can use the following menus and buttons to filter alarms for the ACL currently selected:

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>and/or pull-down menu</td>
<td>Choose the and or or operator to apply to this row of the filter definition. This operator is present only in the second and subsequent rows.</td>
</tr>
</tbody>
</table>
Edit ACLs Dialog

<table>
<thead>
<tr>
<th>Menu (Blank) / Not Pull-Down Menu</th>
<th>Choose <strong>not</strong> in order to search for all systems except those that meet this row of the filter definition. Otherwise, leave blank.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Criterion Pull-Down Menu</td>
<td>Choose the criterion to filter for, such as <strong>Severity</strong>, <strong>Hostname</strong>, <strong>Origin</strong>, and so on.</td>
</tr>
<tr>
<td>Operator Pull-Down Menu</td>
<td>Choose the appropriate operator, such as <strong>is</strong>, <strong>contains</strong>, <strong>starts with</strong>, and so on. If you select Severity, the following operators are available: <strong>=</strong>, <strong>&lt;=</strong>, <strong>&gt;=</strong></td>
</tr>
<tr>
<td>Text Field / Alarm Severity-Level Pull-Down</td>
<td>Enter the appropriate text for the criterion you selected. If you select Severity for the criterion, a pull-down menu listing alarm severity-levels is displayed. Choose from the following: <strong>Clear</strong> (0), <strong>Informational</strong> (1), <strong>Warning</strong> (2), <strong>Minor</strong> (3), <strong>Major</strong> (4), <strong>Critical</strong> (5).</td>
</tr>
<tr>
<td>Add Filter / Remove Filter Icons</td>
<td>Click to add or remove rows for the filter definition.</td>
</tr>
<tr>
<td>Move Up / Move Down Icons</td>
<td>Click to move the row up or down. Filter rows are applied in sequential order.</td>
</tr>
</tbody>
</table>

**UMP Views**

The Edit ACLs dialog in Account Admin lets you define ACL permissions to fit your organization's needs. For example, you can define which UMP portlets can be accessed, or whether actions can be taken on alarms.

If an ACL includes the permission to access a given portlet, you may wish to restrict what is visible in the portlet. You can accomplish this by clicking the UMP Views button in the Edit ACLs dialog. This launches the Configure UMP Views window in which you can fine-tune whether certain views or features are visible to account contact users with the selected ACL.

**Example**

If an ACL includes the permission **USM Basic**, users with that ACL have access to USM. By default, they can see the following icons displayed above the navigation tree in USM:
The above icons allow users to select the tree view or one of three badge views. If you select **USM Navigation** in the Configure UMP Views dialog, and then de-select **Show tree view**, only the following badge view icons are displayed to users with the ACL you selected:

![Icons](image)

To restrict certain UMP views, choose an item in the left column of the Configure UMP Views dialog, and de-select corresponding items in the right column. The UMP Views dialog allows you to restrict the following items:

<table>
<thead>
<tr>
<th>USM Navigation</th>
<th>Show tree view</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Show badge 1 view</td>
</tr>
<tr>
<td></td>
<td>Show badge 2 view</td>
</tr>
<tr>
<td></td>
<td>Show badge 3 view</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>USM Tree</th>
<th>Show geo view</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Show inventory</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>USM Group</th>
<th>Show group view</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Show alarms view</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>USM Device</th>
<th>Show system view</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Show alarms view</td>
</tr>
<tr>
<td></td>
<td>Show metrics view</td>
</tr>
<tr>
<td></td>
<td>Show groups view</td>
</tr>
<tr>
<td></td>
<td>Show monitoring view</td>
</tr>
</tbody>
</table>
The Change Password application allows account contact users to change their passwords. You must be an account contact user and have the Change Password permission set in the Nimsoft Access Control List (ACL) in order to use this application.

Administrators (users with the Account Administration permission in the ACL) can use the Account Admin (see page 25) application to change an account contact user’s password or ACL permissions.

To change your password:

1. Enter your existing password in the Old Password field.
2. Enter your new password in the New Password field.
   
   Passwords must be at least six characters long.
3. Enter the new password in the Confirm Password field.
   
   The text in the New Password and Confirm Password fields must match exactly.
4. Click Change Password.
The Admin Console allows you to manage your CA Nimsoft Monitor infrastructure. To view this documentation, see the *Admin Console User Guide* (http://docs.nimsoft.com/prodhelp/en_US/AdminConsole/7.0/index.htm).

This document contains information about:

- How to configure probes
- How to deploy robots
Chapter 5: Alarm Console

The Alarm Console is the main window for viewing and managing alarms. The window displays information about alarms in a table format, and you can use toolbar icons and menu options to view information and take action on alarms.

The Alarm Console is integrated with other CA Nimsoft applications, such as the Enterprise Console and Infrastructure Manager, and it can also be opened in the Service Level Manager.

This section contains the following topics:

- Alarm Console Toolbars (see page 35)
- Alarm Console Table (see page 40)
- Popup Menus (see page 42)
- Dialogs (see page 45)
- Enabling the Alarm Console (see page 55)
- Troubleshooting Alarm Console (see page 55)

## Alarm Console Toolbars

The Alarm Console has a series of four toolbars you can rotate through by clicking the Rotate icon:

- **General Bar** (see page 35)
- **Command Bar** (see page 36)
- **Alarm Filter Bar** (see page 37)
- **History Bar** (see page 38)

### General Bar

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotate icon</td>
<td>Click to display the next toolbar.</td>
</tr>
<tr>
<td>Details button</td>
<td>Opens the Details (see page 48) dialog, where you can view additional information about the alarm and manage the alarm.</td>
</tr>
<tr>
<td>Find button</td>
<td>Click to open the Find (see page 48) dialog, where you can search for alarms that contain specified text.</td>
</tr>
</tbody>
</table>
**Alarm Console Toolbars**

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>View pulldown</td>
<td>Selects the columns to display. Click the word View to return to the Default view.</td>
</tr>
<tr>
<td>Filter field</td>
<td>Enter text to display only alarms that contain that text.</td>
</tr>
<tr>
<td>Column pulldown</td>
<td>Click and select a column to apply the Filter field to that column. For example, if you want to see only alarms from a certain host, enter all or part of the host name in the Filter field and choose Host from this pulldown.</td>
</tr>
<tr>
<td>Severity icons</td>
<td>Click on an icon to display only alarms with that severity. The colors correspond to the following severity levels:</td>
</tr>
<tr>
<td></td>
<td>■ Red = Critical</td>
</tr>
<tr>
<td></td>
<td>■ Orange = Major</td>
</tr>
<tr>
<td></td>
<td>■ Yellow = Minor</td>
</tr>
<tr>
<td></td>
<td>■ Blue = Warning</td>
</tr>
<tr>
<td></td>
<td>■ Light blue = Informational</td>
</tr>
<tr>
<td>Stop updates icon</td>
<td>Click to stop updates to the information in the Alarm Console. Otherwise, the alarm information is updated dynamically.</td>
</tr>
<tr>
<td>Help icon</td>
<td>Click to display the online help for the Alarm Console.</td>
</tr>
</tbody>
</table>

**Command Bar**

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotate icon</td>
<td>Click to rotate to the next toolbar.</td>
</tr>
<tr>
<td>Accept button</td>
<td>Click to assign the selected alarm to yourself.</td>
</tr>
<tr>
<td>Assign button</td>
<td>Click to open the Assign To (see page 47) dialog, where you can assign the selected alarm to another user.</td>
</tr>
<tr>
<td>Unassign button</td>
<td>Click to unassign the selected alarm.</td>
</tr>
<tr>
<td>Acknowledge button</td>
<td>Acknowledges the alarm. The alarm is considered closed and is removed from the Alarm Console table. The alarm is also deleted from the NimBUS Alarm Server (NAS) database, but a copy is retained in the history database.</td>
</tr>
<tr>
<td>Attach button</td>
<td>Click to open the Attach Notes (see page 47) dialog, where you can attach notes to the selected alarm.</td>
</tr>
</tbody>
</table>
Alarm Console Toolbars

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Detach button</td>
<td>Click to open the <a href="#">Detach Notes</a> dialog, where you can detach notes from the selected alarm.</td>
</tr>
<tr>
<td>Actions button</td>
<td>Click to open the <a href="#">Actions</a> dialog, where you can execute an action to launch a URL.</td>
</tr>
<tr>
<td>Severity icons</td>
<td>Click on an icon to display only alarms with that severity. The colors correspond to the following severity levels:</td>
</tr>
<tr>
<td></td>
<td>■ Red = Critical</td>
</tr>
<tr>
<td></td>
<td>■ Orange = Major</td>
</tr>
<tr>
<td></td>
<td>■ Yellow = Minor</td>
</tr>
<tr>
<td></td>
<td>■ Blue = Warning</td>
</tr>
<tr>
<td></td>
<td>■ Light blue = Informational</td>
</tr>
<tr>
<td>Stop updates icon</td>
<td>Click to stop updates to the information in the Alarm Console. Otherwise, the alarm information is updated dynamically.</td>
</tr>
<tr>
<td>Help icon</td>
<td>Click to display the online help for the Alarm Console.</td>
</tr>
</tbody>
</table>

### Alarm Filter Bar

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotate icon</td>
<td>Click to rotate to the next toolbar.</td>
</tr>
<tr>
<td>Alarm Filter pulldown</td>
<td>Choose an alarm filter to apply. Filters display only specified alarms. Click the words Alarm Filter in the toolbar to clear all filters that have been applied.</td>
</tr>
<tr>
<td>Set Alarm Filter button</td>
<td>Click to create, edit, or delete a filter. Opens either the <a href="#">Alarm Filter Expression</a> dialog, if the Default filter is selected, or the <a href="#">Alarm Filter</a> dialog if a filter other than the Default is selected. The Alarm Filter Expression dialog allows you to specify criteria for the filter. The Alarm Filter dialog allows you to choose to add, edit, or delete a filter.</td>
</tr>
</tbody>
</table>
Alarm Console Toolbars

Filter Management button
Click to display a popup menu:
- **Save Current Alarm Filter** - Saves the current alarm filter. A dialog opens where you can enter the name for the filter. Once saved, you can choose the filter from the Alarm Filter pulldown menu.
- **Rename Current Alarm Filter** - Click to rename the current filter. A dialog opens where you can edit the name of the filter.
- **Delete Current Alarm Filter** - Click to delete the current filter.
- **Manage Alarm Filter ACL Associations** - Opens the [Manage Alarm Filter ACLs](#) dialog, where you can set which ACLs the filter is accessible for.

Severity icons
Click on an icon to display only alarms with that severity. The colors correspond to the following severity levels:
- **Red** = Critical
- **Orange** = Major
- **Yellow** = Minor
- **Blue** = Warning
- **Light blue** = Informational

Stop updates icon
Click to stop updates to the information in the Alarm Console. Otherwise, the alarm information is updated dynamically.

Help icon
Click to display the online help for the Alarm Console.

### History Bar

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotate icon</td>
<td>Click to rotate to the next toolbar.</td>
</tr>
</tbody>
</table>
### Alarm Console Toolbars

<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Alarm History button</td>
<td>Click to open the <a href="#">Alarm History</a> dialog, where you can view a list of alarms for the selected time period in a separate window. The list includes acknowledged alarms, which are stored in the Nimsoft Alarm Server (NAS) history database.</td>
</tr>
<tr>
<td>Query button</td>
<td>Click to open the <a href="#">Query</a> dialog, where you can define a filter for the historical alarms you want to see.</td>
</tr>
<tr>
<td>Run Query button</td>
<td>Opens the <a href="#">Run Query</a> dialog, where you can execute a previously defined query.</td>
</tr>
<tr>
<td>Transaction History button</td>
<td>Click to open the <a href="#">Transaction History</a> dialog for the selected alarm. The Transaction History dialog shows the transaction log for the alarm, including the initial message, any suppressions, the closure message (when the alarm is acknowledged), and any other actions. Note: You can only view the transaction history for an acknowledged alarm by clicking the Transaction History button in the Alarm History dialog.</td>
</tr>
<tr>
<td>Severity icons</td>
<td>Click on an icon to display only alarms with that severity. The colors correspond to the following severity levels:</td>
</tr>
<tr>
<td></td>
<td>- Red = Critical</td>
</tr>
<tr>
<td></td>
<td>- Orange = Major</td>
</tr>
<tr>
<td></td>
<td>- Yellow = Minor</td>
</tr>
<tr>
<td></td>
<td>- Blue = Warning</td>
</tr>
<tr>
<td></td>
<td>- Light blue = Informational</td>
</tr>
<tr>
<td>Stop updates icon</td>
<td>Click to stop updates to the information in the Alarm Console. Otherwise, the alarm information is updated dynamically.</td>
</tr>
<tr>
<td>Help icon</td>
<td>Click to display the online help for the Alarm Console.</td>
</tr>
</tbody>
</table>
The Alarm Console table lists information about alarms received. By default, a certain set of columns is displayed. You can change the columns displayed, or display all columns, by using the View pull-down menu on the General toolbar (see page 35). Click the text View next to the pull-down menu to return to the default view.

You can create filters to display alarms with only the specified criteria using the Filter field and the column pull-down menu, or by using the features on the Alarm Filter toolbar (see page 37). Click the word Filter or Alarm Filter to clear any filters applied.

Click on a column header to sort by that column. Clicking the column header again toggles between ascending and descending sort order.

You can change the width of the columns in the Alarm Console table. To do so, click on the column border and drag it to the width you want.

The Alarm Console table contains the following information. The columns you see depends on the setting you choose for the View pulldown menu.

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Severity</td>
<td>Level of seriousness of the alarm. The severity levels are:</td>
</tr>
<tr>
<td></td>
<td>■ Clear = Green</td>
</tr>
<tr>
<td></td>
<td>■ Informational = Cyan</td>
</tr>
<tr>
<td></td>
<td>■ Warning = Blue</td>
</tr>
<tr>
<td></td>
<td>■ Minor = Yellow</td>
</tr>
<tr>
<td></td>
<td>■ Major = Orange</td>
</tr>
<tr>
<td></td>
<td>■ Critical = Red</td>
</tr>
<tr>
<td>Assign icons</td>
<td>An icon appears in this column if the alarm is assigned to a user.</td>
</tr>
<tr>
<td>Notes icons</td>
<td>An icon appears in this column if a note is attached to the alarm.</td>
</tr>
<tr>
<td>ID</td>
<td>An automatically generated unique identification number.</td>
</tr>
<tr>
<td>Host</td>
<td>The name of the computer hosting the probe sending the alarm.</td>
</tr>
<tr>
<td></td>
<td>Note: Occasionally the IP address of the host is displayed in this column</td>
</tr>
<tr>
<td></td>
<td>instead of the name. You may be able to convert the IP addresses to names</td>
</tr>
<tr>
<td></td>
<td>by changing the StripHostName setting in the Windows registry. However, this</td>
</tr>
<tr>
<td></td>
<td>may result in timeouts.</td>
</tr>
<tr>
<td></td>
<td>To change the StripHostName setting, navigate to the setting as follows and</td>
</tr>
<tr>
<td></td>
<td>change the value to 1:</td>
</tr>
<tr>
<td></td>
<td>HKEY_CURRENT_USER &gt; Software &gt; Nimbus</td>
</tr>
<tr>
<td></td>
<td>Software &gt; NimBUS Manager &gt; Options &gt; StripHostName</td>
</tr>
<tr>
<td>Message</td>
<td>Description of the error condition.</td>
</tr>
<tr>
<td><strong>Time Received</strong></td>
<td>The last time the alarm was received by the Nimsoft Alarm Server (nas).</td>
</tr>
<tr>
<td>-------------------</td>
<td>-------------------------------------------------------------------------</td>
</tr>
<tr>
<td><strong>Subsystem</strong></td>
<td>The part of the monitored computer the alarm was sent from (CPU, disk, memory, and so on).</td>
</tr>
<tr>
<td><strong>Probe</strong></td>
<td>Name of the probe that issued the alarm.</td>
</tr>
<tr>
<td><strong>nas</strong></td>
<td>Name of the Nimsoft Alarm Server (nas) storing and managing the alarm.</td>
</tr>
<tr>
<td><strong>Origin</strong></td>
<td>QoS data from probes is tagged with a name to identify the origin of the data. The origin name is set in the controller probe GUI. If the origin name is not set, the hub name is used.</td>
</tr>
<tr>
<td><strong>Count</strong></td>
<td>Number of times the alarm was received from the probe.</td>
</tr>
</tbody>
</table>
| **Visible**       | Whether the alarm is set to Visible or Invisible.  
|                   | = Visible  
|                   | = Invisible  
|                   | To set the alarm to Visible or Invisible, select the alarm and right-click on the Alarm Console table. Choose Set Visible or Set Invisible from the **popup menu** (see page 42). Unless the Invisible Alarms permission is set in the ACL, the user does not see alarms that are marked invisible. To hide invisible alarms, right-click in the toolbar and select **Hide Invisible Alarms**. To show invisible alarms again, right-click in the toolbar and select **Show Invisible Alarms**. |
| **Source**        | The IP address of the computer sending the alarm. |
| **Subsystem ID**  | Identifier made up of one or more numbers separated by dots. This identifies which part of the monitored computer the alarm was sent from (CPU, disk, memory, and so on). |
| **Robot**         | Name of the Nimsoft robot hosting the probe sending the alarm. |
| **Hub**           | The hub controlling the robot hosting the probe that sent the alarm. |
| **Domain**        | The name of the domain associated with the probe sending the alarm. |
| **User Tag 1, User Tag 2** | User-defined tag to be used to group or locate the source of alarms. The tag is set in the controller probe GUI, accessed via Infrastructure Manager. |
| **Custom 1-5**    | Custom columns. You can set the value for these columns in the **Set Custom Fields** (see page 53) dialog. |
| **Time Origin**   | The time when the alarm was sent from the probe. |
| **Time Arrival**  | The time when the alarm was received by the nas. |
| **Assigned To**   | User alarm is assigned to. |
| **Assigned By**   | User who assigned the alarm. |
### Popup Menus

There are two popup menus for the Alarm Console window. Access the popup menus by right-clicking on:

- [The toolbar](#) (see page 42)
- [The Alarm Console table](#) (see page 43)

#### Toolbar Popup Menu

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotate Toolbars</td>
<td>Displays the next toolbar. The Alarm Console has a series of four toolbars you can rotate through. For more information, see <a href="#">Alarm Console Toolbars</a> (see page 35).</td>
</tr>
<tr>
<td>Group Alarms by Severity</td>
<td>Displays alarms of the same severity grouped together in the Alarm Console table.</td>
</tr>
<tr>
<td>Actions</td>
<td>Opens the <a href="#">Actions</a> (see page 45) dialog, where you can execute an action to launch a URL.</td>
</tr>
<tr>
<td>Manage Actions</td>
<td>Opens the <a href="#">Manage Actions dialog</a> (see page 49), where you can create, edit, or delete actions.</td>
</tr>
<tr>
<td>Set Custom Fields</td>
<td>Opens the <a href="#">Set Custom Fields</a> (see page 53) dialog, where you can set the value for the Custom columns in the Alarm Console table.</td>
</tr>
<tr>
<td>Find</td>
<td>Opens the <a href="#">Find</a> (see page 48) dialog, where you can search for alarms that contain specified text.</td>
</tr>
<tr>
<td>Select All Alarms</td>
<td>Selects all alarms in the Alarm Console table.</td>
</tr>
<tr>
<td>Deselect All</td>
<td>Deselects all alarms in the Alarm Console table.</td>
</tr>
<tr>
<td>Invert Selection</td>
<td>Reverses the selection of alarms. Alarms that were selected become deselected, and the unselected alarms become selected.</td>
</tr>
</tbody>
</table>
### Popup Menus

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>View Trace Log</td>
<td>Opens the Trace Log (see page 54) dialog, where you can view information about each transaction for the alarm.</td>
</tr>
<tr>
<td>Preferences</td>
<td>Opens the Preferences (see page 51) dialog, where you can choose settings that affect the behavior of the Alarm Console.</td>
</tr>
<tr>
<td>Set Default View</td>
<td>Opens the Set Default View (see page 53) dialog, where you can set which columns are displayed in the Default view.</td>
</tr>
<tr>
<td>Settings</td>
<td>Opens the Adobe Flash Player Settings (see page 46) dialog, where you can choose settings for Adobe Flash Player.</td>
</tr>
<tr>
<td>About Adobe Flash Player 10</td>
<td>Opens an Adobe web page with information about Flash Player 10.</td>
</tr>
</tbody>
</table>

### Alarm Console Table Popup Menu

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Details</td>
<td>Opens the Details (see page 48) dialog, where you can view information about the selected alarm.</td>
</tr>
<tr>
<td>Accept</td>
<td>Assigns the selected alarm to yourself.</td>
</tr>
<tr>
<td>Assign</td>
<td>Opens the Assign To (see page 47) dialog, where you can assign the selected alarm to another user.</td>
</tr>
<tr>
<td>Unassign</td>
<td>Unassigns the selected alarm from the user it was assigned to.</td>
</tr>
<tr>
<td>Acknowledge</td>
<td>Acknowledges the alarm. The alarm is considered closed and is removed from the Alarm Console table. The alarm is also deleted from the NimBUS Alarm Server (NAS) database, but a copy is retained in the history database.</td>
</tr>
<tr>
<td>Show Notes</td>
<td>Opens the Notes (see page 51) dialog, where you can view notes attached to the selected alarm.</td>
</tr>
<tr>
<td>Attach</td>
<td>Opens the Attach Notes (see page 47) dialog, where you can attach existing notes to the selected alarm or create new ones.</td>
</tr>
<tr>
<td>Detach</td>
<td>Opens the Detach Notes (see page 48) dialog, where you can detach notes from the selected alarm.</td>
</tr>
<tr>
<td>Manage Notes</td>
<td>Opens the Manage Notes (see page 49) dialog, where you add, edit, or delete notes.</td>
</tr>
<tr>
<td>Set Visible</td>
<td>You can hide some alarms by marking them as invisible. To hide alarms, first select the alarms you want to hide in the Alarm Console table, then right-click and choose Set Invisible. Unless the Invisible Alarms permission is set in the ACL, the user does not see alarms that are marked invisible.</td>
</tr>
<tr>
<td>Menu Item</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Set Invisible</td>
<td>If the Invisible Alarms permission is set in the ACL, alarms marked as invisible are displayed in the Alarm Console table in grayed out italics. To change an alarm to visible, right-click on the alarm and choose Set Visible.</td>
</tr>
<tr>
<td>Transaction History</td>
<td>Click to open the Transaction History (see page 54) dialog for the selected alarm. The Transaction History dialog shows the transaction log for the alarm, including the initial message, any suppressions, the closure message (when the alarm is acknowledged), and any other actions. Note: You can only view the transaction history for an acknowledged alarm by clicking the Transaction History button in the Alarm History dialog. See the section Alarm History Dialog.</td>
</tr>
<tr>
<td>Set Filter</td>
<td>Opens the Alarm Filter Expression (see page 46) dialog, where you can create filters to display only the alarms with the properties you specify.</td>
</tr>
<tr>
<td>Copy to Clipboard</td>
<td>Copies the selected alarm(s) to the clipboard.</td>
</tr>
<tr>
<td>Settings</td>
<td>Opens the Adobe Flash Player Settings (see page 46) dialog, where you can choose settings for Adobe Flash Player.</td>
</tr>
<tr>
<td>About Adobe Flash Player 10</td>
<td>Opens an Adobe web page with information about Flash Player 10.</td>
</tr>
</tbody>
</table>
Dialogs

- **Actions** (see page 45)
- **Adobe Flash Player Settings** (see page 46)
- **Alarm Filter Expression** (see page 46)
- **Alarm History** (see page 47)
- **Assign To** (see page 47)
- **Attach Notes** (see page 47)
- **Detach Notes** (see page 48)
- **Details** (see page 48)
- **Find** (see page 48)
- **Manage Actions** (see page 49)
- **Manage Alarm Filter ACLs** (see page 49)
- **Manage Notes** (see page 49)
- **New Action** (see page 50)
- **New Note** (see page 50)
- **Notes** (see page 51)
- **Preferences** (see page 51)
- **Query** (see page 52)
- **Run Query** (see page 52)
- **Set Custom Fields** (see page 53)
- **Set Default View** (see page 53)
- **Trace Log** (see page 54)
- **Transaction History** (see page 54)

**Actions Dialog**

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Actions list</td>
<td>Click to select the action you want to execute. To create, edit, or delete actions, right-click on the Alarm Console toolbar and choose Manage Actions (see page 49) from the popup menu.</td>
</tr>
<tr>
<td>Execute button</td>
<td>Executes the selected action.</td>
</tr>
</tbody>
</table>
Dialogs

Adobe Flash Player Settings Dialog

This dialog allows you to choose settings for Adobe Flash Player. Click the Help button for information about these settings.

Alarm Filter Dialog

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>New</td>
<td>Opens the Alarm Filter Expression (see page 46) dialog, where you can specify criteria for a filter.</td>
</tr>
<tr>
<td>Edit</td>
<td>Opens the selected filter expression for editing.</td>
</tr>
<tr>
<td>Delete</td>
<td>Deletes the selected filter expression.</td>
</tr>
</tbody>
</table>

Alarm Filter Expression Dialog

The Alarm Filter Expression dialog allows you to create a filter for the alarms displayed in the Alarm Console table. You can filter on information in any of the columns in the Alarm Console table. The operator pulldown menu for most fields can be set to = or != (not equal). The Severity, Count, and Notes fields have additional choices for the operator menu. For a description of each field, see Alarm Console Table (see page 43).

Enter the information you want to filter on and then click OK. The Alarm Console table displays only the alarms that match the filter until you modify or clear the filter.

To save the filter expression in the repository click Filter Management, Save Current Alarm Filter on the Alarm Filter toolbar. Then you can choose the filter from the Alarm Filter pulldown menu.

Note: Unlike in Infrastructure Manager, the filters in the Alarm Console table in UMP use only regular expressions, not simple wildcards. For example, to filter for all hubs with a name starting with HUB, use the regular expression /HUB.*/. Entering HUB* will not work.
### Alarm History Dialog

The Alarm History dialog displays information about historical alarms. To access the dialog, click Alarm History on the History toolbar (see page 38), then select a time period. For information about the columns displayed in the table, see Alarm Console Table (see page 43).

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Filter field</td>
<td>Enter text to display only alarms that contain that text.</td>
</tr>
<tr>
<td>Transaction History button</td>
<td>Click to launch the Transaction History dialog, which shows the transaction log for the alarm, including the initial message, any suppressions, the closure message (when the alarm is acknowledged), and any other actions. Launching the Transaction History dialog from within the Alarm History dialog (rather than from the History toolbar or from the Alarm Console table pop-up menu) allows you the additional ability of viewing the transaction log for acknowledged alarms.</td>
</tr>
<tr>
<td>Show all columns check box</td>
<td>Click to show all columns of the Alarm Console. Otherwise, only the columns for the Default view are shown.</td>
</tr>
</tbody>
</table>

### Assign To Dialog

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Assign To list</td>
<td>Lists the users you can assign the alarm to. Click a user name to select it.</td>
</tr>
<tr>
<td>Assign button</td>
<td>Assigns the alarm to the selected user.</td>
</tr>
<tr>
<td>Filter field</td>
<td>Enter text to display only user names that contain that text.</td>
</tr>
</tbody>
</table>

### Attach Notes Dialog

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Attach Notes list</td>
<td>Lists the available notes. Click on the name of a note to select it.</td>
</tr>
<tr>
<td>OK</td>
<td>Attaches the selected note to the alarm.</td>
</tr>
</tbody>
</table>
Dialogs

New Note | Click to open the New Note (see page 50) dialog, where you can create a new note for the alarm.

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Detach Notes list</td>
<td>Lists the available notes. Click on the name of a note to select it.</td>
</tr>
<tr>
<td>OK</td>
<td>Click to detach the selected note from the alarm.</td>
</tr>
</tbody>
</table>

Details Dialog

The Details dialog provides a way to view the information in the Alarm Console table (see page 43) one alarm at a time. To access the dialog, select an alarm in the Alarm Console table and click Details on the General toolbar (see page 35); or right-click and choose Details from the popup menu (see page 43).

The information displayed is the same as in the Alarm Console table. The radio buttons on the left allow you to select which columns to view, similar to the View pulldown menu on the General toolbar. The arrows allow you to view the previous or next alarms in the Alarm Console table.

For more information about the columns or buttons in the Details dialog, see Alarm Console Table (see page 43) and Alarm Console Toolbars (see page 35).

Find Dialog

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Find field</td>
<td>Enter the text you want to search for.</td>
</tr>
<tr>
<td>Column pulldown</td>
<td>Choose the column you want to search for the specified text.</td>
</tr>
<tr>
<td>Select button</td>
<td>Selects the alarms with matching text.</td>
</tr>
<tr>
<td>Find Previous button</td>
<td>Displays the previous instance of the text.</td>
</tr>
<tr>
<td>Find Next button</td>
<td>Displays the next instance of the text.</td>
</tr>
</tbody>
</table>
Manage Actions Dialog

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name/URL list</td>
<td>Click to select the Name/URL of the action you want to modify.</td>
</tr>
<tr>
<td>New button</td>
<td>Click to open the New Action (see page 50) dialog, where you can create an action.</td>
</tr>
<tr>
<td>Edit button</td>
<td>Click to modify the selected action.</td>
</tr>
<tr>
<td>Delete button</td>
<td>Deletes the selected action.</td>
</tr>
</tbody>
</table>

Manage Alarm Filter ACLs

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Alarm Filter list</td>
<td>Click to select the alarm filter you want to set ACL access for.</td>
</tr>
<tr>
<td>ACL list</td>
<td>Click to select the ACLs you want to have access to the selected alarm filter.</td>
</tr>
<tr>
<td>Selected filter is accessible to all ACLs check box</td>
<td>Click to allow all ACLs to have access to the selected alarm filter.</td>
</tr>
<tr>
<td>OK button</td>
<td>Saves changes and closes the dialog.</td>
</tr>
<tr>
<td>Cancel button</td>
<td>Closes the dialog without saving changes.</td>
</tr>
</tbody>
</table>

Manage Notes Dialog

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ID</td>
<td>An automatically generated unique identification number.</td>
</tr>
<tr>
<td>Name</td>
<td>Enter the name of the note.</td>
</tr>
<tr>
<td>Category</td>
<td>Enter the name of a category for the note.</td>
</tr>
<tr>
<td>Auto Remove</td>
<td>Whether or not Auto Remove is enabled. When enabled, the note is automatically removed from the repository when there are no active alarms that are attached to the note.</td>
</tr>
<tr>
<td>Time Modified</td>
<td>Time when the note was created or last modified.</td>
</tr>
</tbody>
</table>
New Action Dialog

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name field</td>
<td>Enter a name for the action.</td>
</tr>
<tr>
<td>URL field</td>
<td>Enter a URL for the action to launch.</td>
</tr>
<tr>
<td>Arguments button</td>
<td>Displays a list of substitution values you can use in the URL. For example, you can use $ASSIGNED_TO and the name of the user assigned to the alarm you are executing the action for is inserted in the URL.</td>
</tr>
<tr>
<td>OK button</td>
<td>Saves the action and closes the dialog.</td>
</tr>
<tr>
<td>Cancel button</td>
<td>Closes the dialog without saving changes.</td>
</tr>
</tbody>
</table>

New Note/Edit Note Dialog

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name field</td>
<td>Enter a name for the note.</td>
</tr>
<tr>
<td>Category pulldown</td>
<td>Choose a category for the note.</td>
</tr>
<tr>
<td>Text field</td>
<td>Enter text for the note.</td>
</tr>
<tr>
<td>Auto Remove check box</td>
<td>Check this to automatically remove this note from the repository when there are no active alarms that are attached to the note.</td>
</tr>
<tr>
<td>OK</td>
<td>Click to save the note. Your user name and the current time are automatically added to the note.</td>
</tr>
<tr>
<td>Cancel</td>
<td>Click to close the dialog without saving changes.</td>
</tr>
</tbody>
</table>
Notes Dialog

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Notes field</td>
<td>Text of the note.</td>
</tr>
<tr>
<td>OK</td>
<td>Saves any changes and closes the dialog.</td>
</tr>
<tr>
<td>Cancel</td>
<td>Closes the dialog without saving changes.</td>
</tr>
<tr>
<td>Add Comment</td>
<td>Click to add a comment to the note.</td>
</tr>
</tbody>
</table>

Preferences Dialog

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Use extended color scheme check box</td>
<td>If selected, the table cells in the Alarm Console table are colored according to severity level.</td>
</tr>
<tr>
<td>Launch on alarm image click pulldown</td>
<td>Choose a dialog to open when the icon in the severity column is clicked:</td>
</tr>
<tr>
<td></td>
<td>• Details (see page 48)</td>
</tr>
<tr>
<td></td>
<td>• Attach Notes (see page 47)</td>
</tr>
<tr>
<td></td>
<td>• Actions (see page 45)</td>
</tr>
<tr>
<td>Immediate client side update of alarm lines check box</td>
<td>If selected, the Alarm Console table updates as alarms are received.</td>
</tr>
<tr>
<td>Use time zone for pulldown</td>
<td>Choose the device you want to use as the source for setting the time zone.</td>
</tr>
<tr>
<td>Set default alarm filter pulldown</td>
<td>Choose the alarm filter to be applied to the Alarm Console table by default upon opening.</td>
</tr>
<tr>
<td>OK</td>
<td>Saves changes and closes the dialog.</td>
</tr>
<tr>
<td>Cancel</td>
<td>Closes the dialog without saving changes.</td>
</tr>
</tbody>
</table>
**Query Dialog**

The Query dialog allows you to search the historical database for alarms from a specified time period. You can also specify information for each field to search for. For information about each field, see Alarm Console Table (see page 43).

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>From Date</td>
<td>Beginning of the time period you want to view alarms for.</td>
</tr>
<tr>
<td>To Date</td>
<td>End of the time period you want to view alarms for.</td>
</tr>
<tr>
<td>OK</td>
<td>Executes the query.</td>
</tr>
<tr>
<td>Cancel</td>
<td>Cancels the query and closes the dialog.</td>
</tr>
<tr>
<td>Reset Query</td>
<td>Clears the fields in the dialog.</td>
</tr>
<tr>
<td>Save Query</td>
<td>Saves the query to a local repository. To execute a query in the repository, click the Run Query button on the History toolbar (see page 38).</td>
</tr>
</tbody>
</table>

**Run Query Dialog**

The Run Query dialog allows you to execute a previously defined query. To access the dialog, click the Run Query button on the History toolbar (see page 38).

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Query list</td>
<td>Select the query you want to run.</td>
</tr>
<tr>
<td>Run Query</td>
<td>Executes the selected query.</td>
</tr>
<tr>
<td>Delete Query</td>
<td>Deletes the selected query from the repository.</td>
</tr>
</tbody>
</table>
Set Custom Fields Dialog

This dialog allows you to set values for the alarm table columns My Custom 1 through My Custom 5. To access this dialog, right-click on the toolbar in the Alarm Console and choose Set Custom Fields from the popup menu.

You must have the appropriate permissions set in the ACL in order to access this dialog.

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Custom radio buttons</td>
<td>Select the custom field you want to set the value for. These fields are displayed as columns in the Alarm Console table.</td>
</tr>
<tr>
<td>Value field</td>
<td>Enter a value for the custom field.</td>
</tr>
<tr>
<td>Set Value button</td>
<td>Click to save changes and close the dialog.</td>
</tr>
</tbody>
</table>

Set Default View Dialog

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Available Columns list</td>
<td>Lists the columns of the Alarm Console table. Click to select a column to add to or remove from the Visible Columns list.</td>
</tr>
<tr>
<td>Visible Columns list</td>
<td>Lists the columns that will be visible in the Default View for the Alarm Console table. These are the columns the user will see unless they select a different view.</td>
</tr>
<tr>
<td>Add button</td>
<td>Adds the selected column to the Visible Columns list.</td>
</tr>
<tr>
<td>Remove button</td>
<td>Removes the selected column from the Visible Columns list.</td>
</tr>
<tr>
<td>Reset button</td>
<td>Resets the Visible Columns list to the default.</td>
</tr>
<tr>
<td>Auto calculation of the alarm list width check box</td>
<td>If selected, the width of the Alarm Console table is set automatically based on the default widths of the columns displayed. If not selected, you can specify a width for the Alarm Console table in the Width spin box. You can adjust the width of individual columns by dragging the column rule in the Alarm Console table. You can also drag the columns to rearrange the order.</td>
</tr>
<tr>
<td>Width spin box</td>
<td>If the Auto calculation of the alarm list width check box is not selected, you can set the width of the Alarm Console table. You can adjust the width of individual columns by dragging the column rule in the Alarm Console table. You can also drag the columns to rearrange the order.</td>
</tr>
</tbody>
</table>
Dialogs

Trace Log Dialog

The Trace Log contains information about actions taken by the Alarm Console. It is primarily used by Nimsoft support personnel to troubleshoot problems with the Alarm Console.

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>Type of item being traced.</td>
</tr>
<tr>
<td>Token</td>
<td>Type of request, action, or incoming event being processed.</td>
</tr>
<tr>
<td>Detail</td>
<td>Additional information about the request, action, or event.</td>
</tr>
<tr>
<td>Time</td>
<td>Time the request, action, or event occurred.</td>
</tr>
</tbody>
</table>

Transaction History Dialog

The Transaction History dialog shows the transaction log for the alarm, including the initial message, any suppressions, the closure message (when the alarm is acknowledged), and any other actions.

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Filter field</td>
<td>Enter text to display only alarms that contain that text.</td>
</tr>
<tr>
<td>Show all columns check box</td>
<td>Click to show all columns of the Alarm Console. Otherwise, only the columns for the Default view are shown.</td>
</tr>
</tbody>
</table>
Enabling the Alarm Console

In order to access the Alarm Console application, you must enable the Alarm Details permission in the Access Control List (ACL).

1. In the Infrastructure Manager, select Security > Manage Access Control List.
2. Select the relevant ACL in the Access Control List pane.
3. In the Permissions pane, select the Alarm Details permission, then click OK.

The following permissions affect the information you can view and actions you can take in the Alarm Console:

- Accept
- Unassign
- Invisible Alarms
- Alarm History
- Acknowledge
- Alarm Management
- Assign
- Reassign

For more information about ACL permissions, see the Infrastructure Manager documentation.

Troubleshooting Alarm Console

This section tells you how to troubleshoot common issues with Alarm Console.
$PASSWORD Substitution in Alarm Console

To use $PASSWORD substitution in Alarm Console, you must enable it by adding a key to the wasp.cfg file.

Follow these steps:

1. Locate the wasp.cfg file in the following directory:
   <UMP_installation>/probes/service/wasp/conf
2. Open wasp.cfg in a text editor.
3. In the webapps/alarmconsole section, add the following line:
   enable_password_arg = true
4. Save and close the wasp.cfg file.
5. Restart wasp.

Alarm Console Loses Filters with Internet Explorer

If you launch the Alarm Console portlet from a dashboard in Internet Explorer and later reload the page, Alarm Console loses its filter state and shows all alarms.

No Data Displayed in Alarm Console

Valid for MySQL and Oracle

Symptom:
When I log into UMP as an administrative user, the Alarm Console portlet does not display any data.

Solution:

This can occur in large environments with approximately 4000 or more robots, where MySQL or Oracle is the database provider.

You may be able to fix this issue by editing the following parameters in the <setup> sections of the wasp.cfg and dashboard_engine.cfg files:

- In the wasp.cfg, increase nimpool_timeout from 30 to 90.
- In the dashboard_engine.cfg, increase dynamic_views from 60 to 120.
Chapter 6: CA APM Cloud Monitor

The CA APM Cloud Monitor allows you to access the online service that monitors your sites, servers, and applications 24 hours a day, 7 days a week.

You must be an NMS account contact user to access the Cloud Monitor portlet. Your NMS account must be associated with a Cloud Monitor account, and you must have the required credentials associated with this Cloud Monitor account.

This section contains the following topics:
- Importing the LAR File for CA APM Cloud Monitor (see page 57)
- Cloud Monitor Server Location Configuration (see page 58)
- Configuring Access to the CA APM Cloud Monitor Portlet (see page 59)

Importing the LAR File for CA APM Cloud Monitor

The CA APM Cloud Monitor portlet does not automatically appear in the Add Portlet list. You have to log in to UMP and import the LAR file to access the portlet.

**Important:** The CA APM Cloud Monitor portlet requires the UMP user’s permissions to include the Cloud Monitor ACL.

To import a LAR file:

2. Click on the CloudMonitor.lar link to download the file.
3. Log in to UMP as the user that requires access to the CA APM Cloud Monitor portlet.
   - **Note:** This user must be an account contact user of an account that has a Cloud Monitor account and associated credentials.
5. Select Private Pages > Export/Import > Import.
6. Click Browse to locate the CloudMonitor.lar file on your machine.
7. Click Import.
   - A message appears indicating that the import was successful. You can now click Back to Private Pages to return to UMP.
Cloud Monitor Server Location Configuration

The server location of your Cloud Monitor instance must be added to the wasp probe configuration.

Follow these steps:
1. In Infrastructure Manager select the wasp probe on your hub.
2. Press the <Shift> key as you right-click, and select Raw Configure.
3. In the left pane, expand the webapps folder, and select cloudmonitor.
4. In the right pane, select the url key, and click Edit Key.
5. Enter the URL of the Cloud Monitor server as the value for the url key. For example, enter http://ump.cloudmonitor.nimsoft.com.
6. Click OK, then click OK to exit the configuration screen.
Configuring Access to the CA APM Cloud Monitor Portlet

This section describes how to configure access to the CA APM Cloud Monitor portlet in UMP.

The same login credentials for the CA APM Cloud Monitor portlet are used for each account contact user of an NMS account. However, you must manually edit the wasp configuration so that it can pass through the credentials of the NMS account that the user is associated with. If access is not properly configured, users will see the error message Not all credentials have been configured for this User Account when they attempt to log in.

Follow these steps:

1. Locate the Mid and Token values you will need when you edit the wasp configuration:
   a. Log into your CA APM Cloud Monitor account.
   b. Click the Reports menu and select the Access item.
      The reports access page opens.
   c. At the bottom of the page, look for the table labeled Tokens can be used to provide access on your account via external applications. If there is a token listed, you can use it. If not click the Add token button to create one.
      Note: Keep this page open, as you will need the Mid and Token values found here in step 3.

2. Verify the NMS Account name(s) to be used for CA APM Cloud Monitor integration:
   a. Start Infrastructure Manager and log into NMS.
   b. Click the Security menu > Account Administration. Make a note of the NMS Account name(s).
      Note: This Account must be attached to an ACL that grants permission to Access to CA APM Cloud Monitor portlet. Use the Manage Access Control List on the Security menu to verify this.

3. Modify the wasp configuration:
   a. Open wasp in Raw Configure.
   b. In the left-hand navigation pane, expand the <webapps> section, expand the <cloudmonitor> section, and locate the <accounts> section.
   c. Under <accounts>, create a new section, and name the section using the NMS account name you verified previously.
   d. Add the key mid to the <NMS_account> section, and specify the value you located previously.
   e. Add the key token to the <NMS_account> section, and specify the value you located previously.
4. Create sections as needed under the `<accounts>` section for additional NMS accounts.

5. Restart wasp.
Chapter 7: Dashboard

The Dashboard portlet allows you to create and view dashboards. Dashboards are useful to monitor key metrics at a glance.

Dashboards display data in graphic elements, such as gauges, charts, tables, images, or shapes. These elements can display data from several types of data sources, and can be customized with a wide range of colors, fonts, and sounds.

Regular Nimsoft users who have the Dashboard Designer permission set in their ACL can create, edit, and publish dashboards.

Account contact users with the Custom Dashboards permission can open published dashboards for viewing.

For more information about ACLs and user types, see the help for the Account Admin (see page 25) portlet.

This section contains the following topics:

- Create a Dashboard: Overview (see page 62)
- View Modes (see page 63)
- Prepare the Canvas (see page 63)
- Add a Widget (see page 65)
- Create and Assign a Data Source (see page 82)
- Set Properties for a Widget (see page 91)
- Change the Appearance of a Widget (see page 96)
- Manage Widgets on the Canvas (see page 98)
- How to Use Parameters (see page 101)
- Manage Dashboards (see page 101)
Create a Dashboard: Overview

The following diagram shows the high-level steps to create a dashboard.

How to Create a Dashboard

1. Prepare the Canvas
2. Add Widgets
3. Create and Assign Data Sources
4. Set Properties for the Widgets
5. Save the Dashboard
6. Publish the Dashboard

Complete
View Modes

The Dashboard portlet allows you to view, edit, and publish dashboards.

In Edit mode, you can create a new dashboard or make changes to an existing dashboard. In Live View mode, you can view a dashboard with live data and any properties that have been applied to the dashboard.

Live View mode allows you to preview your dashboard temporarily, and is not the same as publishing a dashboard. Published dashboards are saved and are available for authorized users to view.

When you create or modify dashboards, you will switch between Live View and Edit modes. Typically you make changes to a dashboard in Edit mode, such as assigning a data source or changing the appearance of a widget, then switch to Live View to verify the display of the dashboard.

To switch from Live View to Edit mode, click the (Cog) icon in the top left corner of the dashboard, then click the (Edit Dashboard) icon.

Note: If the (Cog) icon is not displayed, mouse over the dashboard. The (Cog) icon is hidden when the dashboard is viewed with no activity.

To switch from Edit mode to Live View, choose Live view from the Dashboard menu.

Prepare the Canvas

Set up the canvas for working on a dashboard.

You can set the size and grid functions of the canvas, or set the background for the dashboard.

Size the Canvas

Set the canvas to an appropriate size for your dashboard.

Follow these steps:

1. Click the (Canvas Properties) tab.
2. Click Size to expand the menu.
3. Set the Width and Height of the canvas.
Set the Background Properties

Set the color for the dashboard background, add an image as the background, or set the opacity of the background.

Follow these steps:
1. Click the (Canvas Properties) tab.
2. Click Background to expand the menu.
3. Set the properties for the dashboard background:
   - **Background Color**
     Sets the color for the dashboard background. Enter a hex code or click the Background Color field to display a color picker. By default the background is white, hex code #FFFFFF. Use the slider on the right of the color picker to set the opacity for the color.
   - **Background Image**
     Sets an image as the background for the dashboard. For more information, see Set a Background Image (see page 64).
   - **Opacity**
     Sets the opacity of the background. Click the slider to adjust the opacity.

Set a Background Image

You can set a background image for the dashboard.

**Note:** If you want the dashboard to have a background image, use this procedure instead of using an image widget. Using this procedure automatically places the background image at the back of the layers on the canvas, and any actions you take on the canvas do not affect the background image.

Follow these steps:
1. Click the (Canvas Properties) tab.
2. Click Background to expand the menu.
3. Click in the Background Image field to display the Image Gallery. To add an image to the Image Gallery, click Upload Image and browse to the image.
4. (Optional) Click the opacity slider to adjust the opacity of the background image.
Set the Grid Functions

You can turn the grid on or off, or change its color or spacing. You can also turn the Snap to grid function on or off.

Follow these steps:
1. Click the (Canvas Properties) tab.
2. Click Grid to expand the menu.
3. Set the properties for the dashboard grid:
   - **Grid On/Off**
     - Turns display of the grid on or off.
   - **Color**
     - Sets the color for the grid.
   - **Spacing**
     - Sets the spacing between lines of the grid.
   - **Snap to grid On/Off**
     - Turns the Snap to grid function on or off. When turned on, items snap to the nearest grid line as you drag them around the canvas. This is useful for aligning items.

Add a Widget

Widgets are items you can place on the dashboard that display the status of devices you are monitoring. Available widgets include shapes, images, tables, lines, gauges, and charts.

Drag-and-drop widgets onto the canvas to add them to the dashboard. After adding a widget, assign it a data source. You can also set properties for widgets, change their appearance, and connect them with lines.

**Note:** You must be in Edit mode to work with widgets. Switch to Live View mode to view widgets with current data values.
Add a Widget

Add a Shape

Add a circle or rectangle to display it in the dashboard.

The circle and rectangle widgets support the following data sources:

- Alarm
- Dashboard
- Probe
- QOS
- SLA
- SQL

Follow these steps:
1. Click the (Widgets) icon in the right pane.
2. Click General to expand the menu.
3. Drag a circle or rectangle onto the canvas.

You can now assign a data source for the shape, then set any properties you wish to change.

More information:
Set Properties for a Widget (see page 91)
Create and Assign a Data Source (see page 82)

Add a Line

Add a line to indicate a relationship between widgets.

You can easily connect two widgets by drawing a line between anchor points on the widgets. When you move the widgets, the anchored lines move with the widgets.

You can also create a standalone line (not anchored to widgets) by dragging a line widget onto the canvas.

Lines support the following data sources:

- Alarm
- Dashboard
- QOS
- SQL
The line does not display the value of the data source, but the color of the line reflects the value. After assigning a data source to the line, you can create a color map so that the color changes for different thresholds that you set. Select the line on the canvas and go to **Color** under the (Widget Properties) tab to set colors for the line.

You can also play sounds for different data values. For more information, see [Set Sounds for a Widget](#) (see page 94).

**Add an Anchored Line**

Add an anchored line to connect widgets.

**Follow these steps:**

1. Click the icon on the toolbar to turn on line anchor points.
2. Click an anchor point on a widget where you want the line to begin and drag to an anchor point on another widget.

   The line is created.

   To move a line around the canvas, drag its end points or change the **Points** settings under **Line** in the (Widget Properties) tab. You can also multi-select the line and other types of widgets and move them as a group.

3. (Optional) Click a line on the canvas to add a mid-line anchor point.
   
   You can then drag the anchor point to add angles to the line.

   **Note:** You can add angles to a line, but you cannot create a line curved in an arc.

4. Click the icon on the toolbar to turn off line anchor points.
5. Click the line to select it.
6. Click the (Widget Properties) tab.
7. Click **Line** to expand the menu.
Add a Widget

8. Set the properties for the line:
   - **Thickness**
     Sets the thickness of the line.
   - **Points**
     Sets the position of the endpoints (or mid-points) of the line.
   - **Widget**
     Visible if the line is connected to an anchor point on a widget. Sets which
     widget the line connects to. Click to display a pull-down menu.
   - **Anchor**
     Visible if the line is connected to an anchor point on a widget. Sets which
     anchor point on the widget the line connects to. Click to display a pull-down
     menu.

9. (Optional) Add arrows to the line:
   a. Click the On/Off button next to **Arrows** to turn it on.
   b. Click **Arrows** to expand the menu.
   c. Set the properties for the arrow:
      - **Type**
        Defines whether there is a single or multiple arrow heads on the line. If you
        choose **Multiple**, an arrow head is displayed in the center of each line segment.
      - **Direction**
        Sets whether the arrow points right (head) or left (tail).

   You can now assign a data source to the line or change other properties, such as
   adding a shadow or border, or creating a color map.

Add a Standalone Line

Add a standalone line if you want a line that is not anchored to widgets.

**Follow these steps:**

1. Click the (Widgets) icon in the right pane.
2. Click **General** to expand the menu.
3. Drag a line onto the canvas.

   To move a line around the canvas, drag its end points or change the **Points** settings
   under **Line** in the (Widget Properties) tab. You can also multi-select the line and
   other types of widgets and move them as a group.
4. (Optional) Add a mid-point to the line:
   a. Click the icon on the toolbar to turn line end points on.
   b. Click the line on the canvas to add a mid-point.
      You can now drag the anchor point to add angles to the line.
      **Note:** You can add angles to a line, but you cannot create a line curved in an arc.
   c. Click the icon on the toolbar to turn off line anchor points.

5. Click the line to select it.

6. Click the (Widget Properties) tab.

7. Click **Line** to expand the menu.

8. Set the properties for the line:
   **Thickness**
   Sets the thickness of the line.
   **Points**
   Sets the position of the endpoints (or mid-points) of the line.
   **x, y**
   The x fields set the horizontal position of the endpoint, and the y fields set the vertical position. Enter the x and y position for each endpoint or mid-point.

9. (Optional) Add arrows to the line:
   a. Click the On/Off button next to **Arrows** to turn it on.
   b. Click **Arrows** to expand the menu.
   c. Set the properties for the arrow:
      **Type**
      Defines whether there is a single or multiple arrow heads on the line. If you choose Multiple, an arrow head is displayed in the center of each line segment.
      **Direction**
      Sets whether the arrow points right (head) or left (tail).
      You can now assign a data source to the line or change other properties, such as adding a shadow or border, or creating a color map.
Add a Widget

Add an Image

Add an image widget to display an image on the dashboard.

The following types of image files are supported:
- .gif
- .jpg
- .jpeg
- .tiff
- .png

**Note:** To add a background image to the dashboard, use the **Background Image** setting in the (Canvas Properties) tab instead of an image widget. The **Background Image** setting automatically places the image at the back of the layers on the canvas, and actions you take on the canvas do not affect the image.

The image widget supports the following data sources:
- Alarm
- Dashboard
- QOS
- SLA
- SQL

**Follow these steps:**
1. Click the (Widgets) tab.
2. Click **General** to expand the menu.
3. Drag the **Image** widget onto the canvas.
4. Click the image widget to select it.
5. Click the (Widget Properties) tab.
6. Click **Image** to expand the menu.
7. Click in the **Image** field to display the Image Gallery.
8. Click an image to select it.
   
   To add an image to the Image Gallery, click **Upload Image** and browse to the image.
9. Click **Select**.

   The image is displayed in the widget. By default **Stretch to fit** is turned off. To size the image to fit the frame of the widget, click the button to turn this on.

   You can now assign a data source to the image widget or change the widget’s properties, including defining an image map.

**More information:**

Define an Image Map for an Image Widget (see page 93)

---

**Add a Table**

Add a table widget to display data from an SQL data source in a table.

**Note:** You must have an SQL data source configured to use the table widget.

**Follow these steps:**

1. Click the **(Widgets) tab.**
2. Click **General** to expand the menu.
3. Drag the **Table** widget onto the canvas.
4. Click the **(Data Sources) tab.**
5. Click **SQL** to expand the menu.
   
   If you do not see the **SQL** pane, click **(Options), Show advanced data sources.**
6. Drag an SQL data source onto the table widget.
   
   If you do not see any SQL data sources listed, create one. For instructions, see Create an SQL Data Source (see page 88).

   The table is created with the appropriate rows and columns and is populated with data. You can set the properties for the columns, including adding or deleting columns, in the following steps.

7. Click the table to select it.
8. Click the **(Widget Properties) tab.**
9. Click **Table** to expand the menu.
10. Set the **Row Height** for the table.
11. Click **Columns** to expand the menu.
12. Click the plus sign (➕) to add a column to the table.
13. Click the name of the column header to select it.
14. Enter a name for the column header.

15. Click the name of the data field to select it.

16. Choose a data field from the pull-down menu.

17. Select a column by clicking the column header in the table widget or by clicking its row in the list of columns under the **Columns** menu.

18. Set the properties for the column:

   **Width**
   
   Sets the width of the selected column.

   **Renderer**
   
   Sets the format for data display for the selected column. Choose the type of renderer from the pull-down menu. The remaining column properties vary depending on the renderer selected.

   **TEXT:**
   
   **Font Size**
   
   Sets the size of the text font.

   **Color**
   
   Sets the color of the font. Click in the **Color** field to display a color picker. Use the slider on the right of the color picker to set the opacity for the color.

   **Color Map**
   
   Sets different colors for different values. For example, for a column displaying percent, you can set 90 percent to red and all values of 90 percent or greater are displayed in red in the column. Click the plus sign (➕) to add more color settings.
BAR:

Show Value
Turns on or off the display of the numeric value next to the bar in the column.

Minimum
Sets the minimum value to display for the bar.

Maximum
Sets the maximum value to display for the bar.

Bar Color
Sets the color of the bar.

Bar Color Map
Sets different colors for different values. For example, for a bar displaying percent, you can set 90 percent to red and all values of 90 percent or greater are displayed in red in the bar. Click the plus sign (+) to add more color settings.

GAUGE:

Sets the shape of the gauge, semicircle or circle.

Min
Sets the minimum value to display for the gauge.

Max
Sets the maximum value to display for the gauge.

Default Color
Sets the color to use to indicate the position of the gauge.

Color Map
Sets different colors for different values on the gauge. For example, for a gauge displaying percent, you can set 90 percent to red and all values of 90 percent or greater are displayed in red on the gauge. Click the plus sign (+) to add more color settings.
IMAGE:

Default Image
Sets the image to display in the column. Click in the Default Image field to display the Image Gallery. To add an image to the Image Gallery, click Upload Image and browse to the image.

Stretch to fit
Sizes the image to fit the frame of the widget. Click the button to turn this on.

Image Map
Click the plus sign (➕) to add different images to be displayed for different data values. For example, for a column displaying percent, you can set 90 percent to an image of a warning icon and all values of 90 percent or greater are displayed as the warning icon in the column.

19. (Optional) Add more columns to the table and set the properties for each column.

20. (Optional) To delete a column, select the column by clicking the column header in the table or by clicking its row in the list of columns under the Columns menu, then click the trash can icon (Trash) under the Columns menu.

Add Text

Add a text widget to display text on the dashboard.

You can also use a label to place text under a widget. Use a text widget instead of a label if you want to:

■ Use standalone text.
■ Use a different text format, such as date or numeric.
■ Assign a different data source to the text than the widget.
■ Set properties for the text separately from the widget. For example, using a text widget allows you to position the text independently of the widget, or to apply a shadow to the text box instead of the widget.

There are two types of variables you can use in text widgets:

■ $VAR - Enter this to display the current value of the data source
■ Parameters - Use variables you created in the (Parameters) tab. For more information, see How to Use Variables (see page 101).
The text widget supports the following data sources:

- Alarm
- Probe
- QOS
- SLA
- SQL

**Follow these steps:**

1. Click the (Widgets) icon in the right pane.
2. Click **General** to expand the menu.
3. Drag a text widget onto the canvas.
4. Double-click on the text widget and enter text.

   **Note:** You can also enter the text for the widget in the **Default Value** field under the **Text** menu in the (Widget Properties) tab.

   You can modify other properties for the text widget by doing the remaining steps.

5. Click the text widget to select it.
6. Click the (Widget Properties) tab.
7. Click **Text** to expand the menu.
8. Choose the text format you want from the **Type** pull-down menu.

   For normal text choose **String**.
9. Set properties for the text:

**Font Size**
Sets the size of the text.

**Word Wrap**
Turns the word wrap feature on or off. When turned on, text that is too wide for the text widget is displayed on multiple lines. When turned off, text that is too wide for the text widget is displayed on a single line with an ellipsis (…) indicating that not all text is visible.

**Text Vertical Alignment**
Defines whether the text appears at the top, middle, or bottom of the text box.

**Text Color**
Sets the color of the text. Enter a hex code, or click in the **Text Color** field to display a color picker. Use the slider on the right of the color picker to set the opacity for the color.

**Background Color**
Sets the background color for the text widget. Enter a hex code, or click in the **Background Color** field to display a color picker. Use the slider on the right of the color picker to set the opacity for the color.

10. (Optional) Set a background image for the text widget:
   
   a. Click **Background Image** to expand the menu.
   
   b. Click in the **Image** field to display the Image Gallery.
   
   c. Click an image to select it.
   
   To add an image to the Image Gallery, click **Upload Image** and browse to the image.
   
   d. Click **Select**.
   
   The image is displayed in the text widget. By default **Stretch to fit** is turned off.
   To size the image to fit the frame of the widget, click the button to turn this on.

You can now assign a data source to the text widget if wanted or change the widget’s properties, including defining a color map.
Add a Gauge

Add a gauge widget to display a semi-circular or circular needle gauge in the dashboard.

The gauge widget supports the following data sources:
- Probe
- QOS
- SLA
- SQL

Follow these steps:
1. Click the (Widgets) icon in the right pane.
2. Click Gauges to expand the menu.
3. Drag a gauge onto the canvas.
4. Click the gauge to select it.
5. Click the (Widget Properties) tab.
6. Click Gauge to expand the menu.
7. Set the properties for the gauge:

   - Sets the shape of the gauge, semicircle or circle.
   - Sets the minimum value for the gauge.
   - Sets the maximum value for the gauge.
   - Performs calculations on the data before displaying the value. For example, a value reported in bytes per second can be displayed as bits per second by setting the Unit Multiplier to 8 and the Unit Divisor to 1.
   - Sets the color to use to indicate the position of the gauge.
Add a Widget

Color Map
Sets different colors for different values on the gauge. For example, for a gauge displaying percent, you can set 90 percent to red and all values of 90 percent or greater are displayed in red on the gauge. Click the plus sign (+) to add more color settings.

Decimal Places
Sets the number of decimal places displayed on the gauge. Add decimal places to see finer grained measurements. For example, for a metric that has a minimum value of 0 and a maximum value of 5, you may want to add a decimal place so that the gauge needle displays a more precise position. However, for percentages, with a minimum value of 0 and a maximum value of 100, you may want to leave the decimal places set 0.

You can now assign a data source to the gauge or change properties such as the label, size and position, and so on.

Add a Linear Gauge
Add a linear gauge to display a thermometer-style gauge. This gauge can be vertical or horizontal.

The linear gauge widget supports the following data sources:
- Probe
- QOS
- SLA
- SQL

Follow these steps:
1. Click the (Widgets) icon in the right pane.
2. Click Gauges to expand the menu.
3. Drag a linear gauge onto the canvas.
4. Click the linear gauge to select it.
5. Click the (Widget Properties) tab.
6. Click Gauge to expand the menu.
7. Set the properties for the linear gauge:

**Vertical**
Defines whether the gauge is vertical or horizontal. Turn on for a vertical
gauge, off for a horizontal gauge.

**Ticks Visible**
Defines whether ticks (lines) indicating unit intervals are shown.

**Labels Visible**
Defines whether labels for the ticks are shown.

**Labels Size**
Sets the size of the numbers on the gauge.

**Labels Offset**
Sets how far from the ticks, and on which side, the numbers are displayed.

**Min**
Sets the minimum value for the gauge.

**Max**
Sets the maximum value for the gauge.

**Unit Divisor, Unit Multiplier**
Performs calculations on the data before displaying the value. For example, a
value reported in bytes per second can be displayed as bits per second by
setting the Unit Multiplier to 8 and the Unit Divisor to 1.

**Bar Thickness**
Defines the width of the bar that indicates the value.

**Bar Offset**
Sets where the bar is displayed.

**Tick Size**
Defines the vertical length of the ticks.

You can now assign a data source to the linear gauge or change properties such as
its label, size and position, and so on.
Add a Line Chart

Add a line chart to display a graph of QOS data in the dashboard.

The line chart widget displays data series for one or more QOS data sources.

Note: You must have a QOS data source configured in order to use the line chart.

Follow these steps:
1. Click the (Widgets) icon in the right pane.
2. Click Charts to expand the menu.
3. Drag a line chart onto the canvas.
4. Click the chart to select it.
5. Click the (Widget Properties) tab.
6. Click Chart to expand the menu.
7. Set the properties for the chart:
   - X-Axis Label
     Displays a label for the x- (horizontal) axis. The x-axis displays the time interval for the chart.
   - Y-Axis Label
     Displays a label for the y- (vertical) axis. The y-axis displays the unit for the QOS measurement.
   - Series Duration in Hours
     Sets the number of hours of data displayed, ending with the current hour. For example, if it is 10:15 and you set this to 4 hours, data from 6:00 to 10:00 is displayed.
   - Min Value
     Sets the minimum value to display on the y-axis.
   - Max Value
     Sets the maximum value to display on the y-axis.
   - Timezone
     Sets the time zone for the chart.
8. Click Series to expand the menu.
9. Click the plus sign (+) to add a data series to the chart.
10. Click the name of the data series to select it.
11. Enter a name for the data series.
12. Set the properties for the data series:

**Series Data Source Type**

Sets the type of data source for the selected data series. Only the QOS data source is supported.

**Series Data Source Name**

Sets the data source for the selected data series. If no data sources are listed in the pull-down menu, create the QOS data sources for the chart. For instructions, see [Create a QOS Data Source](see page 85).

**Display Type**

Sets the type of chart to display for the selected data series: area, line, or plot.

**Color**

Sets the color for the selected data series. Click in the Color field to display a color picker. Use the slider on the right of the color picker to set the opacity for the color.

**Size**

Sets the size of the line or plot points for the selected data series.

**Drop Shadow**

Turns on or off the display of a shadow below the selected data series.

**Unit Multiplier, Unit Divisor**

Performs calculations on the data before displaying the value. For example, a value reported in bytes per second can be displayed as bits per second by setting the Unit Multiplier to 8 and the Unit Divisor to 1.

13. (Optional) Create additional data series.

14. (Optional) Use the up and down arrows to move the selected data series up or down in the list.

If data for multiple data series overlaps, the series highest in the list is displayed. For example, if you have a data series measuring CPU usage for the user and a data series measuring CPU usage for the system, and the values for both are 0 for the time interval, the data for the data series that appears first in the list is displayed.

You can now change other properties for the line chart, such as its label or size and position.
Create and Assign a Data Source

In order for widgets to display data, assign a data source. First create a data source, then drag-and-drop the data source onto widgets.

A single data source can be assigned to multiple widgets. Data sources are available only for the dashboard you create them for.

Available data source types are:

- Alarm
- Dashboard
- SLA/SLO
- QOS

There are also advanced data sources that require special knowledge, such as familiarity with your CA Nimsoft Monitor environment or with creating database queries. Advanced data sources are:

- Probe
- SQL

Note: Not all widgets support all data sources. When you drag a data source onto a widget, the widget border turns green if the widget supports the type of data source, red if it does not.
Create and Assign a Data Source

Create an Alarm Data Source

Create an alarm data source if you want widgets to display a color indicating the highest severity alarm that meets specified criteria.

To specify the criteria, create a filter that defines which alarms to include.

This filter is in addition to alarm filters defined for the user’s ACL (access control list) in the Account Admin portlet.

For example, if an account contact user is restricted in their ACL to seeing only alarms from a particular hub, only these alarms are included in alarm data displayed by Dashboard widgets. In the Dashboard portlet, the administrator creating the dashboard can further limit the alarm data displayed for particular widgets, such as including only alarms from a particular subsystem. This is done by defining a filter when you create an alarm data source.

**Note:** Alarm filters for ACLs may also be defined in Infrastructure Manager, but these filters are not applied in the Dashboard portlet. We recommend using the Account Admin portlet to manage accounts, users, and ACLs.

**Follow these steps:**
1. In the right pane, click the (Data Sources) tab.
2. Click Alarm, then click the (plus sign).
3. Enter information in the fields of the dialog to create the filter:
   - **Name**
     - Enter a name for the data source.
   - **(blank) or not**
     - Choose not in order to search for all systems except those that meet this row of the filter definition. Otherwise, leave this field blank.
   - **Select a field for filtering**
     - Choose the criterion to filter for, such as Hostname, Source, Message, and so on.
   - **Select operator**
     - Choose the appropriate operator, such as is, contains, starts with, and so on.
   - **Text field**
     - Enter the appropriate text for the criterion you chose. You can enter regular expressions in this field.
   - **Plus/Minus Sign**
     - Click to add or remove rows for the filter definition.
4. Click Test Filter.

Alarms that match the filter are listed. Verify the results are as expected and adjust the filter if necessary.

5. Click Create.

Create a Dashboard Data Source

You can use a dashboard with alarm data sources as a data source for another dashboard. Create this type of data source if you want to drill down from one dashboard to another. The parent dashboard displays the highest severity alarm from the child dashboard.

Note: The dashboard data source is supported only for dashboards with alarm data sources.

Use this functionality to logically group elements together, emphasizing the geographical, topographical, structural, or organizational placements of monitored systems.

For example, you have a dashboard named Regions with widgets that show the alarm status for North America, Asia, Europe, Africa, and Latin America. You create a child dashboard named North America that shows the alarm status for West Coast, Midwest, and East Coast. In the Regions dashboard, you assign the North America dashboard as a data source for the North America widget. When you see an alarm status of major for North America in the Regions dashboard, hover over the North America widget and click the link to the North America dashboard in the tooltip. This opens the North America dashboard, where you can see which area of North America generated the major alarm.

Note: The tooltip with the drill down link for a dashboard data source is displayed only in Live View or in a published dashboard. It is not displayed in Edit mode.

Save the parent and child dashboards with the same visibility setting (private, public, or account). For example, if you save the parent dashboard as an account dashboard and the child dashboard as private, account contact users cannot drill down from the parent to the child because they do not have permission to view the child dashboard. Save the child dashboard again and change the visibility setting to match the parent dashboard, then re-publish the child dashboard.

If you change the visibility setting after publishing a dashboard, you must re-publish it in order for the new visibility setting to take effect.

Follow these steps:

1. In the right pane, click the (Data Sources) tab.
2. Click Dashboard.

Your dashboards are listed as data sources.
Create an SLA or SLO Data Source

You can use a Service Level Agreement (SLA) or Service Level Objective (SLO) as a data source. Create this type of data source if you want widgets to display the compliance percentage of the SLA or SLO.

For more information about SLAs and SLOs, see the help for the SLA Reports (see page 293) portlet.

Follow these steps:
1. In the right pane, click the (Data Sources) tab.
2. Click SLA/SLO.
   Your SLAs and SLOs are listed as data sources.

Create a QoS Data Source

Create a QoS data source if you want widgets to display the current value for a QoS metric.

You can display QoS metrics stored in the Nimsoft Information Store (NIS) database, or any database that has a connection configured in the jdbc tab of the wasp configuration file.

The wasp (Web Application Service Provider) is an embedded Tomcat web server running as a probe. It is distributed to the system during the UMP installation, and afterward, appears as a probe in Infrastructure Manager or the Admin Console. For more information about configuring a database for wasp, see the wasp probe guide in the CA Nimsoft Product Information Library.

Follow these steps:
1. In the right pane, click the (Data Sources) tab.
2. Click QOS, then click the (plus sign) icon.
3. Enter information in the fields of the dialog:
   
   **Name**
   Name for the data source.
   
   **QoS**
   Select the QoS.
   
   **Source**
   Select the source for the QoS metric. Typically the source is the system where the probe is running.
   For example, for a CPU usage metric the source is the system where the cdm probe is installed.
   
   **Target**
   Select the target for the QoS metric. Typically the target is the system being monitored.
   For example, for a CPU usage metric the target is the source where CPU usage is measured. For a URL response metric the target is the endpoint of the measurement.
   
4. Click **Test QoS (Single)** to view the latest value for the QoS, or **Test QoS (Multi)** to view values for the last hour.
   
   **Test QoS (Multi)** is useful to preview data for the line chart widget.
   
   After clicking the **Test QoS** button, you see the value and date for the metric.
   
5. Click **Create**.

---

**Create a Probe Data Source**

Create a probe data source if you want widgets to display data from a specific probe. Each probe has a set of commands you can execute by creating a probe data source.

For example, use the `get_info` command for the cdm probe to return a variety of information about the system, including OS, uptime, and CPU usage.
Note: The probe data source is an advanced data source and requires knowledge of the probe command set and parameters. The probe command set can be explored using the Probe Utility. For more information about the Probe Utility, see the Admin Console User Documentation or the Infrastructure Manager User Guide, available in the CA Nimsoft Product Information Library.

Follow these steps:
1. In the right pane, click the (Data Sources) tab.
2. If Probe is not listed as a data source, click the (Options) icon and choose Show advanced data sources.
3. Click Probe, then click the (plus sign).
4. Enter information in the fields of the dialog:
   - Name
     Enter a name for the data source.
   - Hub
     Choose the hub for the probe.
   - Robot
     Choose the robot where the probe is installed.
   - Probe
     Choose the probe.
   - Command
     Choose the probe command to collect the data you want to display. For example, if you want to display CPU status information from the cdm probe, choose cpu_status. The cpu_status command collects a number of metrics related to CPU status, such as cpu idle, processor queue length, cpu wait, and so on.
   - Parameters
     If the probe command has parameters, enter them here.
   - Results
     Ignore this field until after the next step.
5. Click Test Probe.
   The metrics collected by the command you selected are listed in the Results table.
6. In the Results table, choose the metric that you want widgets to display.
   Only results with a single value are supported. Results that return multiple objects are not supported by the probe data source.
7. Click Create.
Create and Assign a Data Source

Create an SQL Data Source

Create an SQL data source if you want widgets to display a metric from a database.

Follow these steps:

1. In Edit mode, click the (Data Sources) tab.
2. If SQL is not listed as a data source, click the (Options) icon and choose Show advanced data sources.
3. Click SQL, then click the (plus sign) icon next to the database you want to query.
4. Enter information in the fields of the dialog:
   - **Name**: Enter a name for the data source.
   - **Type**: Choose *SQL Statement* to enter an SQL query or *Stored Procedure* to execute a stored procedure.
   - **Query**: Enter the SQL query or stored procedure call.
     For a stored procedure call, use the following syntax:
     
     ```
     exec <storedProcedureName> <parameter1> <parameter2>
     ```
     For example, for a stored procedure named getMetrics with two parameters the syntax is:
     
     ```
     exec getMetrics version_param increment_param
     ```
5. Click Test Query.
   Results of the query are displayed. Verify the results and adjust the query if necessary.
6. Click Create.

Add a Database for SQL Data Sources

By default the Nimsoft Information Store (NIS) database is available for creating SQL data sources. You can add other SQL databases to display data stored there.

Follow these steps:

1. In Edit mode, click the (Data Sources) tab.
2. If SQL is not listed as a data source, click the (Options) icon and choose Show advanced data sources.
3. Click **SQL**, then click the + (plus sign) above the list of databases.
   The Create Database Connection dialog is displayed.

4. Enter information in these fields:
   - **Name**
     Name for the database connection.
   - **Driver Class**
     Defines the type of driver to use to access the database. For example, here are some possible settings:
     - `com.microsoft.sqlserver.jdbc.SQLServerDriver`
     - `oracle.jdbc.driver.OracleDriver`
     - `com.mysql.jdbc.Driver`
   - **JDBC URL**
     JDBC connection URL. For example, here are some possible settings:
     - `jdbc:sqlserver://<dbserver>;DatabaseName=<dbname>`
     - `jdbc:oracle:thin:@<dbserver>:<port>:<service name>`
     - `jdbc:mysql://<dbserver>:<port>/<dbname>`
   - **User**
     User name to access the database.
   - **Password**
     Password to access the database.

5. Click **Create**.

6. Restart the wasp probe.
Assign a Data Source

Assign a data source to a widget to display the current value for that data source.

**Note:** Not all widgets support all data sources. When you drag a data source onto a widget, the border around the widget turns green if the widget supports the type of data source, red if it does not.

**Follow these steps:**

1. In the right pane, click the (Data Sources) tab.
   Widgets that do not have a data source assigned are gray and have a missing data source icon overlaid on them, such as this text widget:

2. Drag a data source from the right pane and drop it onto a widget.
   The data source is now assigned to the widget.

3. (Optional) Choose **Live view** from the **Dashboard** menu to view the widget with current data.

4. (Optional) To change the data source, drag a different data source onto the widget.

**Note:** You can also view or assign a data source for a widget under the **Data Source** menu in the (Widget Properties) tab.

Unassign a Data Source

You can unassign a data source if you want no data source assigned to the widget.

**Note:** To change the data source assigned to a widget to another data source, assign a different data source. You do not need to unassign the current data source first.

**Follow these steps:**

1. Click the widget to select it.

2. Click the (Widget Properties) tab.

3. Click **Data Source** to expand the menu.

4. Click the (Remove data source) icon.

   No data source is assigned to the widget. The data source you removed is still available to assign to other widgets.
Delete a Data Source

Delete a data source if you no longer want it to be available to assign to widgets.

You cannot delete a data source that is in use. First make sure the data source is not assigned to any widgets, then delete the data source.

Follow these steps:
1. Click the (Data Sources) tab.
2. Click a data source to select it.
3. Click the (Delete selected data sources) icon

   The data source is deleted and is no longer listed in your data sources.

Set Properties for a Widget

You can set a number of properties for a widget, such as its name, label, position on the canvas, or a URL link.

Depending on the type of widget and the data source assigned, you may be able to set sounds for different data values. For image widgets, you can set images for different data values (an image map).

Position a Widget

Position a widget to set where in the dashboard the widget appears.

Follow these steps:
1. Click and drag the widget on the canvas to position it.
2. (Optional) Refine the position by entering X and Y coordinates.
   a. Click the (Widget Properties) tab.
   b. Click **Size & Position** to expand the menu.
   c. Enter coordinates for **X Position** and **Y Position**.

   **Note:** To move a line around the canvas, drag its end points or change the **Points** settings under **Line** in the (Widget Properties) tab. You can also multi-select the line and other types of widgets and move them as a group.
Set Properties for a Widget

Name a Widget

Name a widget to be able to identify it in your list of widgets in the (Navigator) tab. If you do not enter a name, a default name, such as Circle_0, is assigned to the widget.

Note: The name of a widget does not appear on the dashboard. To display a name under a widget, add a label for the widget.

Follow these steps:
1. Click the widget to select it.
2. Click the (Widget Properties) tab.
3. Click General to expand the menu.
4. Enter the name in the Name field.

Label a Widget

Label a widget to display text under the widget on the dashboard.

Follow these steps:
1. Click the widget to select it.
2. Click the (Widget Properties) tab.
3. Click Label to expand the menu.
4. Enter text in the Label field.
5. (Optional) Change the font size or color, or the background color, by using the appropriate fields.

Set a URL for a Widget

Set a URL for a widget to display a Web page when you click on the widget.

Follow these steps:
1. Click the widget to select it.
2. Click the (Widget Properties) tab.
3. Click General to expand the menu.
4. Enter a fully qualified URL in the URL field.
   For example:
   https://www.myserver.com
Define an Image Map for an Image Widget

You can create an image map for an image widget.

An image map displays different images for different data values. For example, for an image widget with an alarm type data source you can display a different icon for each alarm severity level.

Follow these steps:
1. Click the widget to select it.
2. Click the (Widget Properties) tab.
3. Click **Image** to expand the menu.
4. Set the default image if you have not done so:
   a. Click in the **Image** field to display the Image Gallery.
   b. Click an image to select it.
      
      To add an image to the Image Gallery, click **Upload Image** and browse to the image.
   c. Click **Select**.
      
      The image is displayed in the widget. By default **Stretch to fit** is turned off. To size the image to fit the frame of the widget, click the button to turn this on.
5. Create the image map:
   a. Assign a data source to the image widget if you have not already done so.
      
      **Note:** You can set the default image but cannot define an image map if a data source is not assigned.
   b. Click the plus sign (+) to add image rows.
      
      **Note:** If you assigned the alarm type data source to the widget, a field for each alarm level is displayed. There is no plus sign as you do not need to add image rows.
   c. Click in the image field to display the Image Gallery.
   d. Click an image file to select it.
      
      To add an image to the Image Gallery, click **Upload Image** and browse to the file.
   e. Click **Select**.
Set Properties for a Widget

f. Enter a value in the field to the right of the image field.
   This value functions as a threshold, and when the threshold is met the image is displayed. For example, if you enter 20, the default image is displayed unless the current value for the data source is 20 or greater.

g. Repeat these steps to add as many images and associated data values as wanted.

Set Sounds for a Widget

For most widgets, you can play sounds for different data values.

All widgets except the table and the line chart support sounds. Sounds are not available if a QOS data source is assigned to a widget.

The types of sound files you can play depend on your browser. If you want sounds to play in multiple browser types, consider using a portable format such as MP3 or ogg.

Follow these steps:

1. Assign a data source to the widget if you have not already done so.
2. Click the widget to select it.
3. Click the (Widget Properties) tab.
4. Click Sound to expand the menu.
   Note: You must assign a data source before the Sounds tab is displayed.
5. Click the (+) sign to add a sound row.
   Note: If you assigned the alarm type data source to the widget, a field for each alarm level is displayed. There is no plus sign as you do not need to add sound rows.
6. Click in the lefthand field of the row to open the Sound Gallery.
7. Click an audio file to select it.
   To listen to an audio file, click the (Play sound) icon next to the file name.
   To add an audio file to the Sound Gallery, click Upload Sound and browse to the file.
8. Click Select.
9. Enter a value in the field to the right of the sound field.
   This value functions as a threshold, and when the threshold is met the sound is played. For example, if you enter 20, the sound is played if the value of the data source is 20 or greater.
10. Repeat these steps to add a sound for each threshold level that you want a sound notification for.
Add Links to a Table

You can add links to a table widget so that users can open a URL from the table. This is useful to link to reports that display more detail about the data in the table. For example, if a robot address is listed in the table, you can link to a report with data about the robot.

You can add as many links as you want.

You can also use substitution to use values from the table in the URL. Under the Columns menu in the (Widget Properties) tab you can see a data field listed for each column. The data field is the field in the data source where that column gets its data. To use substitution, include the name of that data field preceded by a dollar sign ($) in the URL. For example, for a data field named date, enter $date in the URL, as in:

http://www.mypage.com/$date

When a user clicks in a cell in the date column of the table, the date in that cell is used in the URL.

Follow these steps:
1. Click the table widget to select it.
2. Click the (Widget Properties) tab.
3. Click Links to expand the menu.
4. Click the (plus sign) to add a links row.
5. Enter a name for the link.
   This name will appear in a pop-up menu when a user clicks on a cell in the table.
6. Enter the URL.
   If the URL is for a page within UMP, such as a Performance Reports Designer report, you can use a relative URL. Otherwise, use a fully-qualified URL (such as http://www.mypage.com).
   To substitute in the URL the value from the table cell where the user clicks, append the name of the column's data field preceded by a dollar sign ($): http://www.mypage.com/$<data field>
   Note: You can view the data field name for the column under the Columns menu in the (Widget Properties) tab.
7. Switch to Live View mode or publish the dashboard.
8. Click in a cell of the table.
9. Click the link name in the pop-up menu.
   The specified URL is displayed in a new window.
Change the Appearance of a Widget

You can change the way a widget looks, including the size, color, border, or opacity, or you can add a drop shadow.

Resize a Widget

Change the size of a widget as needed.

Follow these steps:

1. Hover over the edge of a widget so that the cursor changes to a line with arrows at each end.
2. Click and drag to change the dimension of the widget.
   
   Or, click a corner of the widget (except the lower right corner) and drag it to change both the width and height.
   
   **Note:** Dragging the lower right corner does not change the width and height of the widget. This is so that for small widgets you can click in this area and drag the widget to move it on the canvas.
3. (Optional) Refine the size by entering the number of pixels for the width and height.
   
   a. Click the (Widget Properties) tab.
   b. Click **Size & Position** to expand the menu.
   c. Enter the number of pixels for **Width** and **Height**.

Set the Opacity

Change the opacity to make the widget fainter or darker.

Follow these steps:

1. Click the widget to select it.
2. Click the (Widget Properties) tab.
3. Click **General** to expand the menu.
4. Click the **Opacity** slider to adjust the opacity.

**Note:** If you set the opacity to 0 the widget is invisible. To select an invisible widget, go to the (Navigator) tab and click the name of the widget. You can then go to the (Widget Properties) tab and change the properties of the widget.
Add a Shadow

You can add a shadow around the edges of a widget.

**Follow these steps:**
1. Click the widget to select it.
2. Click the (Widget Properties) tab.
3. Click the On/Off button next to Shadow to turn the shadow on.
4. (Optional) Use the color picker and the Angle, Distance, and Blur sliders to adjust the appearance of the shadow.

   **Note:** Use the slider on the right of the color picker to set the opacity for the color.

Add a Border

You can add a border to a widget.

**Follow these steps:**
1. Click the widget to select it.
2. Click the (Widget Properties) tab.
3. Click the On/Off button next to Border to turn the border on.
4. (Optional) Use the Color and Width fields to adjust the appearance of the border.

Set the Color

You can change the color of some widgets. For some widgets, you can define a color map, where different values are represented by different colors. For example, for a widget with an alarm type data source you can display a different color for each alarm severity level.

**Follow these steps:**
1. Click the widget to select it.
2. Click the (Widget Properties) tab.
3. Click Color to expand the menu.
4. Enter a hex code, or click the Default Color field to display a color picker.

   Use the slider on the right of the color picker to set the opacity for the color.
5. (Optional) Create a color map, if available:
   a. Assign a data source to the widget if you have not already done so.
      
      **Note:** You can set the default color but cannot define a color map if a data source is not assigned.
   b. Click the plus sign (➕) to add color rows.
      
      **Note:** If you assigned the alarm type data source to the widget, a field for each alarm level is displayed. There is no plus sign as you do not need to add any color rows.
   c. Enter a hex code or click in each color field to display a color picker.

### Set the Width of a Line

You can set the width of a line that connects two widgets.

**Follow these steps:**

1. Click the line to select it.
2. Click the (Widget Properties) tab.
3. Click **Line** to expand the menu.
4. Enter a value in the **Width** field.

### Manage Widgets on the Canvas

There are a number of tools to help you manage widgets on the canvas. You can clone widgets, move them to the front or back, or select multiple widgets.

You can also do other common tasks such as aligning widgets, changing the type of widget, or deleting widgets.

### Clone a Widget

Clone a widget to duplicate the widget and its properties.

**Follow these steps:**

1. Click the widget to select it.
   
   **Note:** Select multiple widgets by dragging the cursor around the widgets and releasing.
2. Click the (Clone) icon on the toolbar.
Multi-Select Widgets

You can select multiple widgets. You can then move the widgets as a group or clone or delete all selected widgets.

Follow these steps:
1. In Edit mode, drag the cursor around the widgets on the canvas and release.
   The widgets are selected.
   To move the widgets as a group, click on one of the widgets and drag.

Align Widgets

Align widgets of the same type by using the position settings. This places the widgets in an even line.

Note: Labels for different types of widgets have different offsets from the widgets. If you want to align text below (or above) widgets, use text widgets instead of labels.

Follow these steps:
1. Drag one of the widgets and position the edge where you want the widgets to align.
   To align widgets horizontally, place the bottom edge of the widget where you want the widgets to align. To align widgets vertically, place the left or right edge where you want the widgets to align.
2. With the widget selected, click the (Widget Properties) tab.
3. Click Size & Position to expand the menu.
4. To align widgets horizontally, copy the value in the Y Position field.
   To align widgets vertically, copy the value in the X Position field.
5. Click another widget to select it.
6. Paste the value in the appropriate field, Y Position or X Position.
   The widgets are aligned.
Move a Widget to the Front or Back

You can move overlapping widgets to the front or back on the canvas.

**Follow these steps:**
1. In Edit mode, click the widget you want to move.
2. Do one of the following:
   - To move the widget all the way to the front or back, click **Move to front** or **Move to back** on the toolbar.
   - To move the widget one layer at a time to the front or back, choose **Move forward** or **Move backward** from the pull-down menus on the toolbar.

View a List of Widgets on the Canvas

You can view a list of widgets on the canvas by clicking the (Navigator) tab. Click a widget in the list to select it.

This is useful to select widgets that are layered or positioned close together, or to select invisible widgets (where the opacity is set to 0). Once a widget is selected, you can change its properties in the (Widget Properties) tab.

Widgets selected on the canvas are displayed in bold type in the Navigator tab.

Change the Type of Widget

You may on occasion want to change the type of widget on the canvas, such as changing a gauge to a linear gauge. To change the widget type, delete the widget and add a new one.

Delete a Widget

Delete a widget to remove it from the dashboard.

**Follow these steps:**
1. Click the widget to select it.
   - **Note:** Select multiple widgets by dragging the cursor around the widgets and releasing.
2. Click the (Delete) icon on the toolbar.
How to Use Parameters

You can create parameters (variables) that you can use in various fields in the Dashboard portlet. This allows you to reuse widgets or data sources and update specific information only in the parameter definition.

For example, you have a dashboard with three widgets displaying information about a computer system. You create a parameter named System1 and enter the host name as the value. You enter ${System1} in the Label field in the (Widget Properties) tab for each widget. If the host name changes, or if you want to use the same dashboard for another customer, you can update the parameter value and the host name is updated in all three widget labels.

You can also use parameters when defining a data source. For example, if you have a parameter named Robot1 and the value is the path to the robot, you can enter ${Robot1} in the Robot field for the probe data source.

The text widget also supports the use of parameters.

Follow these steps:
1. Click the (Parameters) tab.
2. Click the Name field and enter a name for the parameter.
3. Click the Value field and enter the value for the parameter.
   The value can be any text, such as a host name, path to a system, and so on.
4. (Optional) To add another parameter, click outside the row you just edited.
   A row is added.
5. Go to Dashboard, Save.
6. In the field where you want to use the parameter, enter the parameter name preceded by a dollar sign and enclosed in curly brackets:
   ${parameter}

Manage Dashboards

You can manage your dashboards by saving, publishing, or deleting them. You can also share dashboards by exporting them and then importing them in another instance of the Dashboard portlet.
Save a Dashboard

Save a dashboard for future use.

Follow these steps:

1. Open the dashboard in Edit mode.
2. Go to Dashboard, Save.
3. Enter a name in the Path field.

   To group dashboards in a folder, enter the folder name before the dashboard name. For example, to save a dashboard named CPU Usage in a folder named Servers, enter the following:

   /Servers/CPU Usage

4. Choose a setting for the Visibility field:

   **Private**

   When the dashboard is published, only you can open and edit the dashboard.

   **Public**

   When the dashboard is published, regular Nimsoft users with the Dashboard Designer permission can view or edit the dashboard. Account contact users cannot view public dashboards.

   **Account**

   Account contact users for the selected account can view published versions of the dashboard. Account contact users can only view dashboards for their account. Accounts are created and managed in the Account Admin portlet.

**Note:** Once you save a dashboard, the icon next to its name in the Open Dashboard dialog indicates its visibility setting:

- ▼ = Private
- □ = Public
- § = Account

If you change the visibility setting after publishing a dashboard, you must re-publish it in order for the new visibility setting to take effect.
Delete a Dashboard

Delete a dashboard you no longer need.

**Follow these steps:**
1. Open the dashboard in Edit mode.
2. Go to Dashboard, Delete.

Publish a Dashboard

Publish a dashboard to save a version of the dashboard for viewing. Publishing creates a copy of the dashboard that is not affected by changes made to the dashboard until it is published again.

For example, if you publish a dashboard and then open the dashboard in Edit mode, add an image widget, and save the dashboard, the published version does not include the image widget. If you publish the dashboard again, the published version is updated to include the image widget. This way you can make changes to the dashboard without affecting the version others may be viewing.

Published dashboards are displayed in a frame with a toolbar. Hover over a toolbar icon to see a description of its function.

**Note:** If you make changes to the dashboard in Edit mode and decide you want to go back to the published version, open the published version and click the Revert icon (↩).

**Follow these steps:**
1. In Edit mode, open the dashboard.
2. Go to Dashboard, Publish.
   The dashboard is published.
3. (Optional) Go to Dashboard, View published version.
   The published version is displayed.

**More information:**

Set Permissions for UMP Portlets (see page 504)
Export a Dashboard

You can export a dashboard that can be imported into another instance of the Dashboard portlet. This allows you to share dashboards with other users.

Follow these steps:

1. Open the dashboard in Edit mode.
2. Go to Dashboard, Export.

   A zip file containing the dashboard files is created and saved in your browser's download location.

   **Note:** You must save changes to the dashboard before exporting it. If Export is not active on the menu save the dashboard.

Import a Dashboard

You can import a dashboard zip file that was exported from the Dashboard portlet.

**Note:** You may need to reassign data sources after importing the dashboard.

Follow these steps:

1. Open the Dashboard portlet in Edit mode.
2. Go to Dashboard, Import.
3. Browse to the dashboard file you want to import and select it.
4. Click Open.

   The dashboard is imported and you can now open it.
Migrate Legacy Dashboards

The Dashboard portlet is new in CA Nimsoft Monitor 7.5. If you had a previous installation of CA Nimsoft Monitor and have dashboards created with the Dashboard Designer portlet, any published dashboards are automatically migrated when you upgrade to CA Nimsoft Monitor 7.5 (specifically when you upgrade to UMP 7.5).

Legacy dashboards that have been saved, but not published, are not migrated.

In most cases no action is needed by you to migrate your published legacy dashboards. However, if there was an error during the migration, when you open the dashboard in the new Dashboard portlet you see a sticky note widget on the dashboard:

Click the sticky note widget to select it, then click the (Widget Properties) tab. Error messages about the migration are displayed in the Notes section.

In most cases when an error occurs a default value is used in place of the property that had the error. Edit the default value as needed.

If you cannot open a migrated dashboard, you can view the messages about migration errors in the portal.log file, located in the ../Nimsoft/probes/service/wasp directory.

Once you install CA Nimsoft Monitor 7.5, we strongly recommend you use the new Dashboard portlet to edit or create dashboards. Changes you make to existing dashboards in the Dashboard Designer will not be reflected when you open the dashboards in the Dashboard portlet because they have already been migrated. If you create and publish a new dashboard in the Dashboard Designer, or change the name of an existing dashboard and publish it, it will be migrated the next time the wasp probe is restarted.

Important! Use the new Dashboard portlet, not the Dashboard Designer portlet, to edit existing dashboards or create new ones. If you edit an existing dashboard in the Dashboard Designer after installing Nimsoft Monitor 7.5, the changes are not reflected in the version of the dashboard you open in the Dashboard portlet because it has already been migrated. If this occurs, change the name of the dashboard in the Dashboard Designer, publish the dashboard, then restart the wasp probe to trigger migration of the dashboard.
Migration of Alarm Widgets

Dashboards created with the legacy Dashboard Designer portlet may include alarm widgets, which are replaced by the alarm data source in the Dashboard portlet.

The alarm filter for the alarm data source, defined when you create an alarm data source, has some differences from the alarm filter for the legacy dashboards alarm widget:

- The alarm data source filter is case sensitive, while the legacy alarm widget filter was not.
- The alarm data source filter does not support the wildcard shortcut notation supported by the legacy alarm widget.
- There may be minor differences in the way regular expressions are interpreted.

Because of this, alarm data sources that were migrated from legacy alarm widgets may behave differently than the legacy alarm widgets. If this happens, modify your alarm filters in the alarm data source.

Follow these steps:

1. In the right pane, click the (Data Sources) tab.
2. Click Alarm to expand the menu.
3. Double-click the data source you need to modify.
4. Modify the alarm filter so that it contains case-sensitive text matches or standard regular expressions.
Chapter 8: Custom Dashboards (Legacy)

This application lists the custom dashboards created using the legacy Dashboard Designer. The Dashboard Designer is replaced in UMP 7.5 with the Dashboard portlet, and dashboards created using the Dashboard Designer are automatically migrated. For more information, see Migrate Legacy Dashboards (see page 105).

The Custom Dashboards portlet is available for existing customers. Which dashboards you see depends on the permissions set in the ACL for your user account.

If you do not have the Custom Dashboards permission set in the ACL, you will see a "Permission Denied" message when you try to run the Custom Dashboards application.

This section contains the following topics:

Dashboard Pane (see page 107)
Managing Alarms in Dashboards (see page 109)
Guest Dashboards Not Displayed (see page 109)

Dashboard Pane

The color of the icons in the tree structure represents the highest alarm severity for alarm objects on the dashboards. Double-click an icon and the corresponding dashboard is launched in the dashboard pane.

The dashboards can contain meters, alarm objects, gauges, charts, tables, panels, and so on.

Alarm and panel objects reflect the severity level of the alarm with the highest severity. Double-clicking an alarm object brings up the alarm list, enabling you to manage the alarms. See Managing Alarms in Dashboards (see page 109) for more information.
Mini Map Tool

The Mini Map tool zooms in on an area of a dashboard. A minimized version of the dashboard is shown in the Mini Map window. A slider lets you zoom in or out of the dashboard on the canvas.

Dragging the yellow field in the Mini Map lets you pan the dashboard.
Managing Alarms in Dashboards

If the dashboard contains alarm objects, you can display the related alarms in the Alarm Console by double-clicking an alarm icon that is not green (there are no alarms associated with green objects).

Guest Dashboards Not Displayed

Valid for multiple-UMP configurations

Symptom:
I have the Superuser ACL, but when I log into UMP, I don’t see all of the Guest dashboards in the Custom Dashboards portlet.

Solution:
This issue may occur in environments with multiple UMP servers if the setting **Allow full dashboard access for Superuser ACL** is not enabled on each instance of the dashboard_engine probe.

To enable this setting, follow these steps:
1. Open Infrastructure Manager, and locate the dashboard_engine probe under the Service node.
2. Double-click on the dashboard_engine probe, and then select the **Advanced** tab.
3. Under the **General** heading, select **Allow full dashboard access for Superuser ACL**.
4. To see all Guest dashboards, repeat steps 1 - 3 on each instance of the dashboard_engine probe.
Chapter 9: Dashboard Designer (Legacy)
The Dashboard Designer portlet is replaced by the Dashboard portlet in CA Nimsoft Monitor 7.5. If you had a previous installation of CA Nimsoft Monitor and have dashboards created with the Dashboard Designer portlet, these dashboards are automatically migrated when you upgrade to CA Nimsoft Monitor 7.5 (specifically when you upgrade to UMP 7.5).

For more information about migration of legacy dashboards to the Dashboard portlet, see Migrate Legacy Dashboards (see page 105).

Once you install CA Nimsoft Monitor 7.5, we strongly recommend you use the new Dashboard portlet to edit or create dashboards. Changes you make to existing dashboards in the Dashboard Designer will not be reflected when you open the dashboards in the Dashboard portlet because they have already been migrated. If you create and publish a new dashboard in the Dashboard Designer, or change the name of an existing dashboard and publish it, it will be migrated the next time the wasp probe is restarted.

**Important!** Use the new Dashboard portlet, not the Dashboard Designer portlet, to edit existing dashboards or create new ones. If you edit an existing dashboard in the Dashboard Designer after installing Nimsoft Monitor 7.5, the changes are not reflected in the version of the dashboard you open in the Dashboard portlet because it has already been migrated. If this occurs, change the name of the dashboard in the Dashboard Designer, publish the dashboard, then restart the wasp probe to trigger migration of the dashboard.

Dashboard Designer allows you to monitor computer systems on your network for QoS data and alarms using various template widgets, such as alarm objects, meter objects, charts, and tables.

- Alarm objects can be filtered to reflect the state of the computers you want to see
- Meter objects can be connected to data sources such as QoS, probes, or variables
- Panels can be used to build dashboards with several levels in a tree structure
- Table objects can be used if you want to present the output from a query to the NIS as a table in a dashboard

The layout of the dashboard components and the background canvas can be configured with a wide range of colors, fonts, sounds and data sources.

You can also import and use pre-made dashboard templates. There are four Dashboard Templates available: two for network devices and two for server systems.

You will also find several general objects, like text objects, images etc. The Dashboard Designer also contains a Preview tool, letting you see the appearance and layout of the Dashboard before publishing it.

The Dashboards will, when saved and published, be available in the Unified Management Portal. There you can see the state and QoS values of the monitored systems and also manage the alarms.
If you do not have the Dashboard Designer permission set in the ACL, you will see a "Permission Denied" message when you try to run the Dashboard Designer application. You must also have the Dashboard Publish permission set in the ACL in order to publish dashboards.

This section contains the following topics:

- [Designing a New Dashboard](#) (see page 113)
- [The Application Window](#) (see page 157)
- [Troubleshooting Dashboard Designer](#) (see page 181)

## Designing a New Dashboard

The following sections describe how to design a new dashboard.

### Create Dashboard

Click the New button to create the new Dashboard.
Designing a New Dashboard

The Create Dashboard dialog appears. Give the Dashboard a name and optionally a brief description. Click the OK button.

When creating a Dashboard, it is mandatory to specify a name, otherwise the OK button at the bottom of the dialog is disabled, and you can’t save the dashboard.
Further validation is done when you click the OK button.

First it checks for invalid characters.

**Note:** The following special characters must NOT be used in dashboard names:
| § ! " # $ % & ' ( ) = ? \ [ ] { } @ £ $ |

If there are no invalid characters, then it checks that the Dashboard name is unique:
**Note:** The Create from Xml option lets you create a Dashboard based on a XML code that you paste into a field appearing in the lower part of the dialog when clicking the Create from Xml option.

The xml code for a Dashboard opened in the Dashboard Designer will be available when clicking the XML button under the Current Dashboard tab.

The xml code will then be opened in a separate window, enabling you to copy the code and use it as input when creating Dashboards in another environment.
Deploying Widgets

You can start deploying objects to the canvas to build your Dashboard. You can also use templates found under the Templates node.

**Note:** You may also save your own Dashboards as templates by right-clicking selecting the components on the canvas and select Save as Template.

Optionally you may prepare the canvas as described in the section Preparing the Canvas if you wish.

Expand the nodes in the Dashboard Components section in the upper left corner to locate the first object you want to use on your Dashboard. Place the cursor on the object, press and hold the left mouse button and drag and drop the object on the canvas.
The object will now appear on the canvas. The red borders indicate that the object is selected. This is important when you have multiple objects on the canvas, so that you know which object to configure.
Now you can set the properties for the object. See the section The Dashboard Components for a description of the properties for the different objects.

When finished with the first object, you may continue with additional objects until your Dashboard is finished.

**Note:** You may easily move, align and distribute objects as described below, see Managing objects on the canvas (see page 134).

When all objects are present, you must save the Dashboard. Click the Save button.

The following dialog appears. Click the OK button to finish the operation.
Finally, click the Publish button to make the Dashboard available.
Using ArrowLine and EllipseLine

ArrowLine enables you to display one or more arrows as directional markers between an alarm line that joins two widgets.

EllipseLine enables you to draw the alarm line as an elliptical curve, rather than just a straight line between the two widgets.

Create a new dashboard by clicking the New button from Dashboard List tab.
Create Dashboard dialog appears. Enter a name for the dashboard. If necessary, you can also enter a description. Click OK button.

From the Dashboard Components, drag and drop the necessary components on the canvas (for example, alarm and gauge widgets).

Next, drag and drop the Alarm Line component.

From AlarmLine Properties section, click the Configure EndPoints button. A dialog appears. Select the checkboxes for the components (in this example, alarm and gauge widgets) and click Apply.
On the canvas, the alarm line links the gauge and alarm widgets.

Choose a thickness for the alarm line by selecting a value from the thickness field.

By default, ellipseX and ellipseY fields are set to 0, hence a straight line is drawn. If you wish to draw the alarm line as an ellipse, specify the necessary parameters in these fields.

Next, click the arrowDirection drop-down button to choose the direction of arrows to be displayed on alarm line. Choose the widget to which you wish to arrow to point to. (In this example, if you choose the alarm widget, the arrow will point from the gauge towards the alarm).
Next you need to specify the number of arrows to be shown on the line. By default, this value is set to 0, hence no arrows are displayed. Enter any number between 0 to 10. If you enter 1, the arrow will be placed at the midpoint of the alarm line. In case of more than one arrow, the arrows will be placed equidistant on the alarm line.

Click Publish button from Current Dashboard tab to save and publish your dashboard.
Using Line Map

Line Map component enables you to display a line that changes its color based on set of data from the external database. You can choose the color of the line when the set of values (such as QoS data) matches a particular data range.

Line Map also supports the ArrowLine (directional markers) and EllipseLine properties discussed in the previous section.

Create a new dashboard by clicking the New button from Dashboard List tab.
Create Dashboard dialog appears. Enter a name for the dashboard. If necessary, you can also enter a description. Click OK button.

From the Dashboard Components, drag and drop the necessary components on the canvas (for example, alarm and gauge widgets).

Next, drag and drop the Line Map component.

From LineMap Properties section, click the Configure Endpoints button. A dialog appears. Select the checkboxes for the components (in this example, alarm and gauge widgets) and click Apply.
On the canvas, the line map links the gauge and alarm widgets.

You can configure the line map properties by specifying the necessary parameters in thickness, ellipseX and ellipseY fields.

Next, click the arrowDirection drop-down button to choose the direction of arrows to be displayed on line map. Choose the widget to which you wish to arrow to point to. (In this example, if you choose the alarm widget, the arrow will point from the gauge towards the alarm).

Next you need to specify the number of arrows to be shown on the line. By default, this value is set to 0, hence no arrows are displayed. Enter any number between 0 to 10. If you enter 1, the arrow will be placed at the midpoint of the alarm line. In case of more than one arrow, the arrows will be placed equidistant on the alarm line.

From the lineColor, choose a color for the arrows displayed on the line map.
Next, click the arrows on the Select Datasource menu bar on the Dashboard Designer toolbar. The Select Datasource dialog appears. In the QoS tab, navigate to and select the required QoS data source. Click OK.

![Select Data Source dialog](Image)

Next, click the Configure Ranges button. Gauge Ranges dialog appears. In the range Start filed, enter the starting value for the range. In the color section, choose the color which you wish to indicate the restive range. You can add more ranges by clicking the + button. To remove a range, select it and then click the – button.

After selecting the colors for all the ranges, click apply.

![Gauge Ranges dialog](Image)
Click Publish button from Current Dashboard tab to save and publish your dashboard.
Converting Data Units

The unit conversion can perform calculations on a data source value for meter and chart widgets before displaying the value.

In the Dashboard section, select the widget that you want to modify the units for.

For example, a value reported in bytes per second can be converted before display to bits per second by setting unitMultiplier to 8 and unitDivisor to 1. See the fields unitMultiplier and unitDivisor in the example below for both Gauge and Chart widgets.
Url Attribute

The widgets Text and Image offer the designer the ability to map a URL to the given widget. If the widget is clicked in player mode, the given URL will be shown in a separate window/tab.
1. Select the image you want to link to a URL.

2. Click Set URL in the Image Properties section. The Set URL dialog is displayed.

3. Enter the URL in the URL text box.
Managing Objects on the Canvas

You can move an object on the canvas by placing the cursor on the object, pressing and holding the left mouse-button, and then drag the objects to a new position.

To be able to work with objects on the canvas (move, align, position, delete, align, set options etc.), the objects must be selected.

You can select multiple objects by placing the cursor on an empty space on the canvas, pressing (and holding) the left mouse button while dragging the cursor over the objects you want to select.

When multiple objects are selected, one of them will automatically be selected as a reference object for alignment of objects, marked with a red border. You can easily select another of them as a reference object for alignment by left-clicking that object.

Preparing the Canvas

It is possible to add a background image or add a background color on the canvas as a background for your Dashboard. You may also decide whether a dashboard panel should be scaled or not when launched.

Selecting the Data Source

Some of the objects used in the Dashboards (meter objects, tables, and trend graphs) display values obtained from a data source. The data can be obtained from:

- QoS data stored in the Nimsoft Information Store (NIS)
- A query to the NIS
- Values from a variable
- A probe
- A database
- An SLA

The Select Datasource dialog is accessed from the Dashboard Designer toolbar. Select the widget for which you wish to configure the data source and click the button on the Select Datasource menu bar.

The Select Datasource dialog is displayed. The dialog has these tabs: QoS, NIS, Variables, Probe, DB and SLA.
After configuring a data source, you can apply it to a widget of your choice by dragging and dropping the icon on that widget, or by clicking Apply.

Not all widgets support each data source, as indicated by this icon:

**Selecting a Data Source—QoS**

The Quality of Service (QoS) data source allows you to access data stored in the QoS database.

**Follow these steps:**

1. Click the Tree radio button.
2. Click the QoS you want, then the Source.
3. If there is more than one Target, click the Target you want.
   
   **Note:** Use the QoS, Source, Target, Probe, and Origin fields to filter for text you enter. For example, if you enter CPU in the QoS field you see only QoS items with CPU in the name.

4. Click Apply, or drag and drop the icon on the object you are assigning the data source to.
Selecting a Data Source--NIS

You can also associate an object with a generic database interface. It’s important to know that the SQL statement should be constructed in such a way that a numeric value will be the result.

If a statement results in several rows of data and even multiple columns, the application will seek for the very first cell, which is the value of the first column in the first row. If using a query as data source for e.g. a table object, it is important that the query must correspond to the number of columns in the table.

Testing the NIS query

Use of long queries or badly written NIS queries can lead to degradation of the system performance. In order to avoid this, you can specify a time-period within which a query is expected to be executed. This value can be set using the sql_query_execute_time key in the dashboard_engine probe. By default, the value for this key is set to 5 seconds.

In case of Gauge widget:

Click the Test button in the Select Datasource dialog for a Gauge widget.

In case of Table widget:

Click the Test button in the Dialog window that appears when you click the Configure Columns button for a Table widget. (Please refer to section Configuring the Table widget for more details about Configure Columns).

When user clicks the Test button, the dashboard_engine executes the query and determines the elapsed time of the query. If the elapsed time of the query is greater than the value specified in the sql_query_execute_time key, the following message appears:

```
Warning

The time taken by SQL Query is 9 seconds which is more than the configured value of 6 seconds. Please change the query else it will result in degraded performance.
```

OK
Selecting a Data Source--Variables

The QoS data can also be obtained from a variable, made available from the variable_server probe. Select the variable from the list and click the OK button.

**Note:** The filter functionality is at the bottom of the dialog. Use the Name Filter field to filter the available data sources. For example, if you enter "gne" in this field, only sources containing the string "gne" in their names will be listed. To reset the filter, click the icon on the right-hand side of the filter field, and all the available data sources are again listed.

Selecting a Data Source--Probe

The Probe data source allows you to select the QoS data from a specific probe.

Start by specifying the Robot where the probe resides from the drop-down list.

Next, select the probe from the drop-down list, which displays all probes found on the specified robot.

The Request drop-down list shows all commands found for the selected probe. For some of the commands, it is possible to add further specifications. Use the Arguments section to add the necessary data.

For some of the probe requests, you can select a result token (the value you want to extract from the data returned by the probe).

**Note:** For some probe requests, the Advanced button is active, and you can use the Advanced button to further refine the result you want.

Finally, specify how often (in seconds) the request is to be executed in the Poll Interval field. The database is typically updated by probes every 5 minutes or so. Check the exact time value by viewing the configuration for the probe in question. Be careful not to specify a poll interval value for the object that queries the database unnecessarily often, as the same data will be collected if an update has not been received.
When you are done click the Test button to verify the value is as expected.

**Advanced button**

This button is active and can be used for certain probe requests.
Click the Advanced button to list all the indexes (each containing a number of result tokens) available for the selected probe request.

Make your choice and click the OK button to save and return to the Select Datasource dialog.
Check the result by clicking the Test button.

Click the OK button to save your data source definition.

Click the Preview button in the Dashboards to see the same value as the Test button in the meter object.

Once saved, the data source is validated again the next time you click Select Datasource button from the same dashboard,

If the selected robot is unavailable, the Robot Address value is indicated by a red border, with a tool tip “Invalid Robot”. The pre-selected probe and its parameters (such as Request, Result Token, and Poll Interval) are retained, thus enabling you to choose a different robot from the list.

In case the selected probe is unavailable, the Probe value is indicated by a red border, with a tooltip “Invalid Probe”. The pre-selected parameters (such as Request, Result Token, and Poll Interval) are retained.

When you select a different probe from the Probe drop-down menu, the pre-configured parameters are cleared.
Selecting a Data Source--DB

The DB data source allows you to execute various SQL commands to fetch any data from the selected database. You can select the database from the Databases drop down list which displays all the databases configured in dap probe. For more details please see dap probe document.

**Note:** The DB tab is available only for Report Template.

To execute stored procedure from dashboard designer dap, perform the following steps:

1. Add any meter, table, chart, or text widget (for e.g. "Gauge", "Table") from the Dashboard Components.
2. Select the widget.
3. Click the arrows on the Select Datasource menu bar.
4. Select the DB tab.
5. Select **dap** from Databases drop down list.
6. To execute stored procedure use the following syntax
7. `exec <sp_name> [ <arg1> <arg2> <arg3> .... <argN> ]`
   
   For example:
   ```
   exec spn_dd_GetQOSDetailsByld 1
   ```
8. Click on Test button.

**Note:** For meter widgets, the stored procedure will display the value of first row and first column provided the value is numeric. For text widget it will display the value of first row and first column.

Selecting a Data Source--SLA

The SLA tab shows a list of Service Level Agreements (SLAs) and their respective Service Level Objectives (SLOs).

When the dashboard is published, the widget shows the compliance percentage of the respective SLA or SLO in the dashboard player.
Configuring the Table Widget

Use the Table widget when you want to present the output as a table in a dashboard.

To select the data source for a Table widget, follow these steps:

1. Click the Table widget to select it.
2. Click the arrows on the Select Datasource menu bar.
   The Select Datasource dialog is displayed.
3. Select and configure the data source, then click OK.
   **Note:** The Table widget supports only NIS, Probe and DB data sources.
4. Click the Configure Column button to configure the table.
5. Click the +- sign in the lower left corner to add rows to the table.
   By default the Table widget displays three rows.

**Header Label**

Replace the entries in the Header Label column with header names describing the selects.

**Data Source**

Replace the entries in the Data Source column with the appropriate data source information.

**Numeric Data**

Check the Numeric Data option for columns with data source returning numeric data.

**Align**

Select the alignment for the values in this row. Valid options are left, middle and right.

**Visualization**

Select the way the result should be presented in the different columns of the table:

- **None**
  The value will be presented as it is returned (as text or a number).

- **Bar**
  The value will be presented as a bar. Default range is 0-100.

- **Gauge**
  The value will be presented as a gauge. Default range is 0-100 Use the Range Min and Range Max to customize the range.
Map

This option lets you specify intervals for how to map numeric values to certain color and text attributes.

Color

Use this palette to select the color of the bar or gauge selected to present the values in this column.

Range Min and Range Max

Gauges and bars will by default have a range from 0 to 100. These ranges can be modified here.

Map

Select this option to enable Set Data button on the right-hand side of the dialog. Click this button to open the Set Map Data dialog.

Here, you can define how the values for the selected items should be presented in the table. You can define a number of ranges, where the values will be presented differently depending on range. Click the ‘+’ button in the lower left corner to add a range. Fill in the range (from and to value), the type (circle, square or text), color and text weight normal or bold for each of the ranges you want.

Auto

Selecting this option, the width of the column will automatically be set to fit the contents displayed in the column.

Width

De-selecting the Auto option (see above), you will be allowed to specify a specific column width in pixels.
Configuring a Widget to Use External DB Data

Some widgets (all meter widgets and the text widget) support showing data from databases other than the NIS database.

The DAP service probe is used to execute the SQL against non-NIS databases and should consequently be active in order to use this functionality.

The values available in the “Databases” drop-down box are configured using the DAP Configurator.
Using Variables for Widgets

You can use variables to assign certain properties to widgets. By using a variable, you can define the properties once and use the variable multiple times. Also, any changes to the properties can be made in a single place.

Variables are shared among all levels of a dashboard (if you use the Panel widget to create multiple levels). However, variables are not shared between dashboards.

There are four types of variables:

QoS
Allows you to specify a QoS measurement. You can specify source, target, probe, and origin for the QoS. This variable can be used with any widget that has a data source.

Robot
Allows you to specify a robot in the Probe tab of the Select Datasource dialog. You can specify the domain, hub, and robot name. This variable can be used with any widget that has a single value. It cannot be used with widgets with multiple values, such as charts (graphs).

Host
Allows you to use a variable for a host name for alarm objects. This variable can be used only in the Host Name field of the Set Alarm Filter dialog.

Text
Allows you to use a variable for text in a text widget.

The following sections describe how to create and use these variables.

More Information:

Variables Pane (see page 181)
Customizing the Dynamic View Templates (see page 188)
Create a Variable

You can create a variable to assign to a widget. This means you can define the properties for a variable once and use it multiple times.

**To create a variable**

1. Expand the Variables node in the tree of the Dashboard Components pane of the Dashboard Designer.
2. Drag-and-drop the type of variable you want onto the canvas.
   A dialog appears.
3. Enter the properties for the variable.
   Fields that are bordered with a red box are required.
   Optional fields that are left blank can be specified per widget. For example, for a QoS variable you can specify the name of the variable and the QoS, and then specify a different Source for each widget the variable is applied to. Fields that are defined for the variable (not left blank) cannot be changed for individual widgets.
   Tip: When creating a QoS variable, you can view a list of QoS measurements by selecting a meter widget on the canvas and then opening the Select Datasource dialog. This may be helpful in getting the exact QoS name.
   For Host variables, the name of the host must be the same as the name displayed in the Alarm Console.
4. Click OK.
   The variable is listed in the Variables pane.

Assigning Variables

The way that you assign a variable to a widget depends on the type of variable it is. The following sections describe how to assign each type of variable.
**Assign a QoS Variable**

You can assign a QoS variable to widgets that have a data source. This allows you to define the properties for a QoS and use it multiple times.

**To assign a QoS variable**

1. Click the widget on the canvas that you want to assign the variable to.
2. Click the arrows on the Select Datasource bar to display the dialog.
3. Click the QoS tab.
4. Click the Variables radio button.
5. Choose the variable from the Select Variable pull-down menu.
6. Enter information in any unpopulated fields.
   - You cannot edit fields that are defined for the variable. To change these fields, edit the variable.
7. Click Apply.
8. Drag-and-drop the data source icon onto the widget on the canvas.

**Assign a Robot Variable**

You can assign a robot variable to widgets that have a single data value, such as a gauge or numeric LED. This allows you to define the properties for a robot once and use it as a source for multiple widgets.

**To assign a robot variable**

1. Click the widget on the canvas that you want to assign the variable to.
2. Click the arrows on the Select Datasource bar to display the dialog.
3. Click the Probe tab.
4. Choose the robot variable from the Robot Address pull-down menu.
5. Enter information in any unpopulated fields.
6. Click Apply.
7. Drag-and-drop the data source icon onto the widget on the canvas.
Assign a Host Variable

You can assign a host variable to an alarm widget. This allows you to enter the name of the host once and use it multiple times.

To assign a host variable
1. Click the alarm widget on the canvas that you want to assign the variable to.
2. Scroll to the bottom of the Alarm Properties pane and click Select Alarm Filter.
   The Set Alarm Filter dialog appears.
3. Click the Value column of the Host Name field and enter $<variable name>.
   Note: You must enter $ before the variable name.
4. Enter information for other fields if wanted.
5. Click OK.

Assign a Text Variable

You can assign a text variable in a text widget. This allows you to enter text once and use it multiple times.

To assign a text variable
1. Click the text widget on the canvas that you want to assign the variable to.
2. Enter $<variable name> in the Text field of the Text Properties pane.
   Note: You must enter $ before the variable name.
   You can use the variable in a phrase, use the same variable multiple times, or use several text variables in one text widget.

Edit a Variable

You can edit a variable to change its properties.

To edit a variable
1. Click the edit icon (📝) next to the variable in the Variables pane.
2. Edit the fields as wanted, then click OK.
Delete a Variable

You can delete a variable.

To delete a variable
1. Click the delete icon (🗑️) next to the variable in the Variables pane.
   If the variable is assigned to an object, you see a confirmation message that lists the object(s) the variable is assigned to.
   **Note:** Deleting a variable that is assigned to objects removes the data source for those objects, and they will not function properly until you assign a new data source.
2. Click Yes to delete the variable, or No to cancel.

Example

This example shows how to create a simple dashboard with a text object and a gauge, showing the disk usage in % for a disk on a computer system.

Launch the Dashboard Designer.

The Dashboard Designer window contains these panes:

**Dashboard Components**
- Widgets that you can drag and drop on the canvas, Image object, Text object, graph object, Table object etc.
- Templates, which are customized objects you can drag and drop on the canvas.

**Dashboards**
- Dashboard List, listing the Dashboards and options for managing Dashboards: Create new, rename, delete etc.
- Current Dashboard, displaying the name of the Dashboard currently on the canvas and listing the objects used on the Dashboard. Tool buttons for copying, renaming, saving, deleting etc.

**Canvas**
- The workspace, on which you drop and configure the objects and create the Dashboards.

**Properties**
- Options for customizing the Dashboard and components on the Dashboard.
Create the new Dashboard by selecting New in the Dashboards section.

Give the Dashboard a name and optionally a description. See the section Dashboard Designer (see page 167) for information about invalid characters in Dashboard names. Click the OK button.
The new (and still empty) Dashboard will appear in the Current Dashboard list.

The Properties window shows the properties options for selected object or Dashboard. Left-clicking the Dashboard or object gives you the possibility to change properties for the object.

So far no objects are added to the canvas, so the properties window shows the properties for the Dashboard background.
You can add a background image or add a background color. In this example we change to a yellow background color.
Above the Properties dialog, you will find four additional properties:

**Snap-To**

If checked, objects dropped on the canvas will snap to the grid. This may be helpful for aligning components when moving them around on the canvas.

**Border**

If checked, all objects on the canvas will have a surrounding box that helps positioning, track and resize the object on the canvas. This border also makes it easier to see which object is selected.

**Grid**

Ticking this option, a grid will appear on the canvas. This makes it easier to align objects etc. The grid will only be visible in design modus.

**Note:** The selected background color, if any, will automatically be changed when selecting the grid option. This is done to make the grid more visible. This change of background color applies for the design modus only. The original background color will be used when the dashboard is stored and published.

**Help**

Selecting this option, the on-line help for the Dashboard Designer will be launched in a separate window.
To add components to the Dashboard, select and drag components from the Dashboard Components window.

The added component has focus, indicated by its red border. The properties window will show this object’s properties, where you can change colors, position, size and data etc.

**Note:** When multiple objects are added, you select an object by left-clicking it, and the object borders change to red. The Properties window will show properties for the selected object, and you can also see the selected object highlighted in the Current Dashboard list.
In this example we will add a text object and a gauge.

**Example:**

Drag a Text object onto the canvas and place it where you like. Left-clicking and holding the mouse button on the object, you can move the object around.
Type My first Dashboard in the text field of the properties dialog. Select text style bold and size 18.

Now, drag a Gauge object, drop it on the canvas and place it below the Text object.

Next, we select a data source for the gauge. Click the arrows on the Select Datasource menu bar to display the Select Datasource dialog. In the QoS tab we select a QoS that gives the percentage of Disk Usage on a computer.
Selecting the ranges in the properties dialog, you can add or remove ranges from the gauge.

If you want to add an additional range from 80-100, you place the cursor in the 80 range and click the + icon in the lower left corner. A new range appears. Type 80 in the Range Start field. Click in the color field and choose a color for the new range.

Click the Apply button in the lower right corner. The gauge will now look like this:
Finally save the dashboard by clicking the Save button.

Click the Preview button to see how the Dashboard appears in the window, and then the Publish button.

Go to the portal window and see that the new Dashboard now appears under the Dashboards node. Left-click it and verify that it appears in the main window.
The Application Window

The Dashboard Designer window contains the following panes:

**Dashboard**

This pane allows you to:
- Create a new dashboard
- Edit an existing dashboard
- Copy an existing dashboard
- Preview a dashboard
- Download dashboard templates to modify and use as your own
- Publish a dashboard
- Generate XML for a dashboard

**Dashboard Components**

This pane contains the building blocks, or widgets (objects), you use when designing dashboards. Drag-and-drop widgets onto the Dashboard canvas area. In addition to widgets, you can use Templates. A template is a widget that you have modified and saved for future use in other dashboards.

**Dashboard History**

This pane, located in the upper right corner, logs changes made to the current dashboard on the canvas. This supports an undo/redo mechanism for a number of operations; typically for adding, deleting, resizing and moving objects.

**Properties**

This pane, located in the lower right hand corner, contains the properties available for the selected object. When designing a dashboard, drag objects from the Dashboard Components section and drop them on the canvas. When you select an object on the canvas, the properties available for the selected object appear in the Properties section. Configure the objects by assigning the properties you want.

**Note:** All the panes can be resized.
Dashboard Components Pane

The following sections describe the Dashboards components available when designing your Dashboards:

- **Alarm Components** (see page 158)
- **Meters** (see page 159)
- **Panel** (see page 160)
- **Dashboard Objects Properties** (see page 163)
- **Selecting the Data Source** (see page 134)
- **Configuring the Table Widget** (see page 140)
- **Scaling Dashboards** (see page 177)
- **Undoing Operations** (see page 166)
- **The Dashboards Templates** (see page 165)
- **Example** (see page 147)

Alarm Components

Use an Alarm object to reflect the alarm state of your monitored computer systems and define what kind of action you want to take place if an alarm with this state occurs. There are two objects available:

- **Alarm**
- **Alarm Line**
**Meters**

Meter objects are well suited when monitoring data sources where the output is a number.

![Dashboard Components](image)

**Gauge**

The Gauge is well suited when monitoring data sources where the output is a number, but where it is not necessary to read a very exact value.

**Numeric LED**

The Numeric LED is well suited when monitoring data sources where the output is a number (free disk space from the cdm probe for example).

**Slider**

The purpose of the Slider is much the same as for the Gauge.

**Note:** Avoid using too short tick intervals when using the Slider object. Otherwise (if using too many ticks), the ticks will not be visible.

**Line Map**

Line Map component enables you to display a line that changes its color based on set of data from the external database. You can choose the color of the line when the set of values (such as QoS data) matches a particular data range.
Panel
Panels may be used to logically group elements together, emphasizing the geographical, topographical, structural or organizational placements of monitored objects.

A Panel object, when placed on the top level of a Dashboard, makes a link to a Panel, one level down in the hierarchy. The Panel object on the top level will reflect propagated alarm states from alarm objects on the level below.

Using the Panel Object

When creating a Dashboard, placing a panel object on the canvas and double-clicking it will bring up an empty canvas one level below. Now you can place and configure other objects on this new canvas.

Clicking the Up button in the Dashboards section brings you up to level 1 again.
Link

Using Link object on a Dashboard, you can create a link from this Dashboard to another Dashboard. To explore the actual linked dashboard, you double-click on the object, and a new window will be loaded with the Dashboard in question. The Link object will reflect propagated alarm states from alarm objects on the linked Dashboard.

Chart & Table

Table

Use this object when you want to present the output from a query to the NIS as a table in a dashboard.

Chart

This object displays the measured values as a graph.

General

Up object

The Up object is used in connection with Panels. Left-clicking the Up object in a Panel brings you one level up in the hierarchy.

Image

The Image object is used to enhance the Dashboard with images of most popular formats (.jpg, .png, .gif, .swf).

Text

Use the Text object for titles, headings etc. to enhance the readability of your Dashboard. Use your own choice of Font, Background and Color.

Connector

Use the connector to draw lines and connect objects. You can define the thickness, color etc.

Report

Time Selector

Use Time Selector to define the duration for which you want to publish the report. You can publish weekly and monthly reports, and also the last day’s report.

In addition, there is a node called Templates. You may save an object as a template. If you have configured an object and want to save and use it in the future when designing other Dashboards, you right-click the object and select ‘Save as Template’.

Configuring the Objects

To select an object on the canvas in order to edit the object properties, click the object. The object borders become red, indicating that the object is selected. The properties dialog for the object appears in the right side of the window.
Dashboard Objects Properties

Dashboard objects are associated with properties that determine the appearance and layout of the object, such as positioning on the canvas, size, text type, shadow effects, or sounds to be played when an alarm occurs.

To select an object on the canvas in order to edit its properties, click the object. The object borders turn red, indicating the object is selected. The properties dialog for the object appears in the right pane.

Some objects, such as meter objects, tables, and trend graphs, are connected to a data source.

Below is an example, describing the properties for an Alarm object.

The Alarm object reflects the alarm state for monitored computer systems. The object reflects the highest severity level of alarms.

**Note:** The ACL settings, defined in Infrastructure Manager, can restrict the alarms the user is allowed to see.

**Object Name**

Each object dropped on the canvas is assigned a unique object name of the form `<object type><object number>`, for example: AlarmObject-D8837BE7-409E-C6DB-2593-C970A9513B7B.

**x and y**

The object’s horizontal and vertical position on the canvas. Position the object by dragging it on the canvas, or by entering values in these fields.

**Width and Height**

The width and height of the object. Change the size of the object by clicking and pausing on one of the edges, then dragging, or by entering values in these fields.

**Alpha**

Transparency of the object. Enter values from 0 to 1, where 0 is most transparent.

**Drop Shadow Enabled**

Adds a shadow effect to the object borders.

**Caption Text**

Label for the object.

**Caption Style**

Color, style (normal or bold), and text size of the caption.

**Caption Text Align**

Select the alignment for the text, centered or left- or right-justified.
Configure Images

Opens the Alarm Images Configuration dialog, where you can add an image to the object. The image is displayed when an alarm occurs.

Single Image

A single image is used, regardless of alarm level.

Alarm Based Images

Allows you to specify different images for the different alarm levels.

Color Masking Image

Allows you to select an image to add as a new layer, masking the color of the object.

Configure Sounds

Opens the Alarm Sound Configuration dialog, where you can select a sound to be played when an alarm occurs.

One Sound

A single sound is used, regardless of alarm level.

Alarm Based Sounds

Allows you to specify different sounds for the different alarm levels.

Select Alarm Filter

Allows you to configure alarm objects to reflect only:

- Alarms from selected host names, source(s), subsystems, and so on
- Alarms containing specific message text
- Alarms assigned to or by specific users at specific times
- Alarms from a specific origin or tagged with user tags

Value

Enter the filter criteria in the Value row for the corresponding parameter(s), then click OK.

To enter a regular expression, place it between two forward slashes. For example: /.*anything goes.*/.  

Note: The dot, asterisk (.* ) combination matches any sequence of characters in regular expressions. A character followed by the asterisk (*) matches 0 or more occurrences of the preceding character. For example, a* matches a, aa, aaa, and so on. The asterisk alone is not a valid character in regular expressions. For more information about regular expressions supported, see the ECMAScript edition 3 language specification (ECMA-262).

If the value is not surrounded by forward slashes, alarms are checked for text that matches the value.
Clear

Clears the filter.

Propagate

Allows the alarm object to propagate its highest severity up to next level in the dashboard hierarchy, if any. (You can create dashboards on multiple levels, using the Panel object.)

The Dashboards Templates

You may save an object as a template. If you have configured an object and want to save and use it in the future when designing other Dashboards, you right-click the object and select ‘Save as Template’.

The object will then be saved under the Templates node, sorted in the group the object belongs to: Alarm objects are placed in the Alarm Components group, meter objects in the Meters group etc.
Undoing Operations

The Dashboard History available when designing dashboards contains functionality for logging changes made to the current dashboard on the canvas. This supports an undo/redo mechanism for a number of operations; typically for adding, deleting, resizing and moving objects.

Clicking the Undo button will undo the operation highlighted in the Dashboard History window.
The Dashboard history will still be present, and you may redo operations removed by
Undo by clicking the Redo button.

The Dashboard History will be cleared when the Dashboard is saved.

Dashboard Designer

The Dashboard Designer provides the following support:

Supported report templates

Dashboard Designer supports three templates: server, interface traffic, and
connectivity with names _rtdv_server, _rtdv_interface and _rtdv_connectivity

The chart widget

The chart objects allows for time period to not be specified and set x-axis based on
time period menu choice in player.

The TimeSelector widget

A new dashboard widget, TimeSelector is implemented. It is a part of dashboard
definition but only used by DashboardPlayer.

Text widget

The Text widget has two attributes:

- Datetime: true/false
- Timezone: local, db and utc (as for chart)

The flex client, if Datetime=true, modifies the time by Timezone and localize the
presentation based on the browser setting.
Dashboard Pane

The Dashboard pane contains a top bar and two tabs, each containing a number of tool buttons.

The top bar

The top bar of the Dashboard pane contains four tool buttons.

Up button

The leftmost button is the Up button, which is activated and can be used only when the active/visible sub panel has a parent panel.
Selecting the Up button when on a Dashboard on for example level 3 in the structure will move you to the Dashboard on level 2. Pressing the Up button again will move you to the Dashboard on level 1.

**Export button**

Click this button if you want to export a Dashboard file (a .zip file). This is convenient if you want to distribute a Dashboard to be used at another portal site. You will be asked if you want to open the file or to save it to disk. Save the file to disk. You may then copy the file to a removable storage device and bring it to the remote portal site and import the dashboard there.
Note: In Mozilla Firefox while downloading for the first time if you have clicked the check box Do this automatically for files like this from now on, then every time you click the Export button it will automatically download in the pre-defined folder.

To enable this dialog box tools > option in Mozilla and click the check box Show the Downloads window when downloading file

Note: The Flash module is not allowed to access the local file system, so therefore a browser link is invoked to complete the save operation.

Import button

Use this button to import existing Dashboards and Dashboard templates.

You may either import a Dashboard from the NIS or from a storage device.
Click the Import Selected Dashboard button to import a Dashboard from the NIS. When new Dashboards are created, they must be published in order to be processed by the Dashboard Engine. The dashboard will then be stored in the Nimsoft Information Store (NIS).

Click the Import Dashboard File button to import a Dashboard from a storage device. This is relevant if you have exported a Dashboard file at another portal site (see above) and want to make it available at your portal site.

**Reload button**

The rightmost button is the reload button. Clicking this button reloads the Dashboard list.

**The Dashboard List tab**

Selecting this tab, all Dashboards designed will be listed.

**Note**: These Dashboards will not be available from the portal before they are published (see the Current Dashboard section).
The following fields and tool button are available:

**Filter field**

This field lets you filter the contents of the Dashboard list. Typing e.g. NIMSOFT in this field, only Dashboards containing the phrase NIMSOFT in the name will be listed. Clear the filter by clicking the x in the right part of the field to clear the filter, and all Dashboards will be listed again.

**Sort A-Z**

Clicking this button will list the dashboards in alphabetical order from A to Z. Clicking the button again will list the Dashboards in the opposite order (from Z to A).

**New**

Click this button to create a new Dashboard. A dialog appears, asking you to specify a name and optionally a brief description of the Dashboard you are going to create.

An additional option, Create from XML, lets you create the new Dashboard based on an existing XML code by pasting the code into the dialog. For details, see the section Designing a new Dashboard (see page 113).

**Open**

Click this button to open an existing Dashboard. Selecting a Dashboard in the Dashboards list and clicking this option, a small dialog appears, asking you to confirm that you want to load the selected Dashboard.

Clicking Yes, the Dashboard will be opened on the canvas.

**Copy**

If you want to make a copy of an existing Dashboard, you select the Dashboard in the Dashboards list and click this button. The following dialog appears. Give the copy a name and click the OK button.
Click the OK button on the next dialog popping up.

On clicking Yes, the dashboard will be opened in Dashboard Designer. On clicking No, the Dashboard will be stored and appear in the Dashboards list.

**Delete**

This option lets you delete the selected Dashboard. When clicking the button, you will be asked to confirm that you really want to delete the Dashboard.

Select “Also delete published version of dashboard” option, to delete the Dashboard from the Nimsoft Information Store (NIS) if the Dashboard has been published.

**Note:** If a client has a dashboard active in the portal and you, as a designer, delete the dashboard, the client will get the following error message and see that the dashboard has been removed from the list of dashboards in the left frame in the portal:

Error in getIncrementalData

The client should accept the message and select another dashboard for viewing.

**The Current Dashboard tab**

Selecting this tab, the selected Dashboard will appear on the canvas. All objects used in the dashboard will be shown under the tab.

The following tool buttons are available:
Save
Saves the selected Dashboard. This can be either a new Dashboard or a modified existing Dashboard.

Edit
This option lets you edit the description of the selected Dashboard.

Publish
Click this button to publish the selected dashboard. When published, the dashboard will be stored in the Nimsoft Information Store (NIS) and will be processed by the Nimsoft Dashboard Engine.

Preview
Clicking this button, the current dashboard will be opened in a separate window. This is useful if you want to see how the Dashboard appears before publishing it and making it available.

View XML code
Clicking this button, the XML code for the selected Dashboard will be available in a separate window popping up. By copying this code and saving it on a storage device, you can use the code when creating Dashboards in another environment.

Properties Pane
The following sections describe background properties.

Background Color
Clicking in the Background Color field in the Properties section, a color palette will be available, letting you choose a color background for your dashboard.
Background Alpha

This option lets you set the background color more or less transparent. This option makes it easy to tune the selected background color. 0 is the maximum transparency.
**Background ImageURL**

You may also add an image on the canvas. Clicking the Select File field, a dialog will be opened, enabling you to choose an image. Select an existing image (one of the ones already uploaded in the Dashboard Designer module). If the image you need is not uploaded, click the Upload New Image tab, browse to locate the file and upload it.

Click the Select Image button to place the selected image on the canvas. The two buttons in the lower left corner of the dialog lets you list the uploaded images either as thumbnails or as a list.
To remove an image from the canvas, you simply click the Delete button (see below).

Scaling Dashboards

The Background Properties pane for the dashboard contains an option called scaleToFit. If you select this option, the dashboard is resized to fit the browser window.

The scopeWidth and scopeHeight parameters let you define the size of the frame to be scaled. This frame appears as a grey rectangle on the canvas. Dashboard components within this frame are handled as a single image and are scaled up or down to match the size of the browser window.

If the scaleToFit option is not selected, the dashboard is displayed at its original size, regardless of the size of the browser window.
General Properties

Above the Properties dialog, you will find four properties:

**Snap-To**

If checked, objects dropped on the canvas will snap to the grid. This may be helpful for aligning components when moving them around on the canvas.

**Border**

If checked, all objects on the canvas will have a surrounding box that helps positioning, track and resize the object on the canvas. This border also makes it easier to see which object is selected.

**Grid**

Ticking this option, a grid will appear on the canvas. This makes it easier to align objects etc. The grid will only be visible in design modus.

*Note*: The selected background color, if any, will automatically be changed when selecting the grid option. This is done to make the grid more visible. This change of background color applies for the design modus only. The original background color will be used when the dashboard is stored and published.

**Help**

Selecting this option, the on-line help for the Dashboard Designer will be launched in a separate window.

*Note*: When selecting an object on the canvas, the properties dialog will be displayed on the right-hand side with the object type displayed on the top bar of the properties dialog.
Right-Clicking Objects on the Canvas

Right-clicking objects on the canvas, a menu pops up with a number of properties:

- **Copy Object(s)**
  - Selecting this option, you can right-click on the canvas and select Paste to paste a copy of the object(s) selected.

- **Save as Template**
  - Selecting this option, the selected object will be saved as a template and can be used when creating other Dashboards. The following dialog appears. Give the template a name and click the OK button.
  - The template will appear in the Dashboard Components section in the Templates > General folder.
Delete Object(s)

Deletes the selected object(s) from the Dashboard. You will be asked to confirm that you really want to permanently delete the object(s).

Bring to Front

Use this option if you have multiple objects covering each other on the canvas, and you want to bring the selected object to front.

Bring Object Forward 1 Step

Use this option if you have multiple objects covering each other on the canvas, and you want to bring the selected object one step forward.

Bring Object Back 1 Step

Use this option if you have multiple objects covering each other on the canvas, and you want to bring the selected object one step back.

Move Object to Back

Use this option if you have multiple objects covering each other on the canvas, and you want to bring the selected object to the background.

Align Horizontal

This option is valid if more than one object is selected. The objects selected will be aligned horizontally. The reference object (the object with red borders) determines the aligned position of the objects. You may select another of the selected objects as reference object by left-clicking it.

Align Vertical

This option is valid if more than one object is selected. The objects selected will be aligned vertically. The reference object (the object with red borders) determines the aligned position of the objects. You may select another of the selected objects as reference object by left-clicking it.

Space Evenly Horizontal

This option is valid if more than two objects are selected. The objects selected will be evenly spaced horizontally.

Space Evenly Vertical

This option is valid if more than two objects are selected. The objects selected will be evenly spaced vertically.

Settings

Opens a small dialog, enabling you to modify the Adobe Flash Player settings.
About Adobe Flash Player

Opens a link to the Adobe home page.

Variables Pane

The Variables Pane lists the variables you have created for the current dashboard.

The pane contains the following fields:

Name

Name of the variable.

Type

Type of variable.

Edit icon (蜇)

Opens a dialog where you can edit the properties for the variable.

Delete icon (删)

Deletes the selected variable.

More Information:

Using Variables for Widgets (see page 143)

Troubleshooting Dashboard Designer

This section tells you how to troubleshoot common issues with Dashboard Designer.
Dashboard API on Windows Does Not Work on Linux

**Symptom:**

When I run the API commands “dashboard.addWidget” or "widget.create" on a Linux robot they do not work correctly.

**Solution:**

Run these commands on a Windows platform.
Chapter 10: Dynamic Views

The Dynamic Views portlet displays automatically generated dashboards for the monitored computer systems and devices on your network. The dashboards display QoS information such as memory, CPU utilization, etc. The data in the dashboards updates at regular intervals, configured in the dashboard_engine probe.

This section tells you how to view the Dynamic View dashboards, how to view Performance Reports of the dashboard data, how to customize the Dynamic View templates, and how to troubleshoot Dynamic Views.

This section contains the following topics:

View a Dynamic View (see page 183)
Understanding the Dynamic Views Tree (see page 184)
Data Sources for Dynamic Views (see page 185)
View System Information in USM (see page 187)
MiniMap Tool (see page 187)
Viewing the Data as a Performance Report (see page 188)
Customizing the Dynamic View Templates (see page 188)
Troubleshooting Dynamic Views (see page 192)

View a Dynamic View

To view a Dynamic View dashboard, double-click a computer system or device in the tree structure. The dashboard is displayed in the Dashboard pane with data for the last 24 hours.

If the dashboard contains alarm objects, you can display the related alarms in the Alarm Console by double-clicking an alarm icon that is not green (there are no alarms associated with green objects).
Understanding the Dynamic Views Tree

The tree structure displays discovered devices in two main groups:

**Network**

Network-related systems, such as network interfaces, routers, and so on. These systems are grouped into two subgroups, Connectivity and Interface Traffic.

**Server**

Server-related systems. By default these systems are grouped into subgroups based on the OS, such as Unix and Windows. Additional groups created in the Unified Service Manager portlet (USM) are also displayed.

The colored icons of the tree nodes represent the highest severity of alarms for the computer systems under that node of the tree. The highest severity level is propagated up the tree structure. For example, if a Windows system sends a critical alarm (red), the icon representing that computer system is red, and the Windows and Server nodes are also red.

To filter the systems you see in the tree, click the Search icon (🔍) and enter text in the text field. Only the items with text that matches are displayed. The filter is applied only to items currently displayed in the tree. For example, if you want to see only Windows servers named with the prefix "Customer1", expand the Server and Windows nodes of the tree so that all Windows servers are displayed. Click the Search icon and enter Customer1 in the text field. Only servers with names that contain "Customer1" are displayed.

When you add or modify groups in USM, the changes are reflected in the Dynamic Views tree; however, it may take several minutes for the Dynamic Views tree to reflect the changes.

Empty groups are not shown in the Dynamic Views tree. There must be at least one system collecting QoS data in the group in order for the group to appear in the tree.

If there are systems that are being monitored that have not been assigned to a group, you see a special group named (Not Grouped). You may want to assign these systems to a group in USM to more easily manage monitoring templates for them.

You can expect to see more systems listed in USM groups than you see in the groups in the Dynamic Views tree. In USM, all discovered systems are listed in the group. The Dynamic Views tree displays only those that are monitored; if they are not monitored, there is no QoS data to generate a Dynamic View.
Data Sources for Dynamic Views

The Dynamic Views tree displays systems that are being monitored. A system is monitored if a probe is collecting QoS data for the system.

The following probes must be collecting data in order for systems to appear in the Dynamic Views tree:

- **Network > Interface Traffic** node: interface_traffic probe
- **Network > Connectivity** node: net_connect probe
- **Server** node: cdm (local) or rsp (remote) probe

These probes all support service-oriented configuration (SOC). With SOC probes, if you selected the Discovery module during CA Nimsoft Monitor installation the computer systems detected on the network will send QoS data as soon as their state is set to Managed. This state can be set in the Unified Service Manager (USM) portlet.

If the Discovery module was not selected during CA Nimsoft Monitor installation, the cdm probe must be distributed to each computer system to be monitored and then configured to send QoS data. Also, the interface_traffic and net_connect probes on the computer running the CA Nimsoft Monitor software must be configured with profiles for each of the systems to be monitored.

Customize Data Sources for Systems

By default the colors of the icons in the Server section of the Dynamic Views tree reflect the status of alarms from the cdm and rsp probes only. However, the icons in the Alarms pane of the Dynamic View dashboard reflect alarms from other probes. For example, the Windows node in the tree may be green, indicating no alarms, but the Availability icon in the Alarms pane of a server dashboard may be red, indicating a critical alarm. This is because the status of the Availability icon reflects alarms from the hub probe.

If you want the alarm status of the tree node icons and the Alarms icons to match, you can add probes as alarm sources for the tree node icons.

To add probes to the alarm status icons, you add a key to the configuration for the dashboard_engine probe. Alarms from each probe listed in the key are reflected in the alarm status of the tree icons.

The following table lists the probes whose alarms are reflected in the Alarms icons.
Note: The icons displayed in the Alarms pane of a Dynamic View dashboard vary depending on which probes are monitoring the system. For example, systems monitored by the rsp probe have fewer Alarms icons than those monitored by the cdm probe.

<table>
<thead>
<tr>
<th>Alarm icon</th>
<th>Probe</th>
</tr>
</thead>
<tbody>
<tr>
<td>Availability</td>
<td>hub</td>
</tr>
<tr>
<td>CPU</td>
<td>cdm or rsp</td>
</tr>
<tr>
<td>Memory</td>
<td>cdm or rsp</td>
</tr>
<tr>
<td>Processes</td>
<td>processes</td>
</tr>
<tr>
<td>Services</td>
<td>ntservices</td>
</tr>
<tr>
<td>Storage</td>
<td>cdm or rsp</td>
</tr>
<tr>
<td>Log Files/Event Logs</td>
<td>logmon, ntevl (Windows)</td>
</tr>
<tr>
<td>All</td>
<td>All probes</td>
</tr>
</tbody>
</table>

Follow these steps:

1. Use Infrastructure Manager to open the Raw Configure window for the dashboard_engine probe:
   a. Select the dashboard_engine probe.
   b. Press the Ctrl key and right-click on the dashboard_engine probe.
   c. Choose Raw Configure.
2. Select the data folder.
3. Click New Key.
   The New Key dialog is displayed.
4. Enter the following values:
   - Key = server_alarm_probes
   - Value = rsp, cdm, <probes you want to add>
   The entries for the key value must be a comma-separated list. Make sure to include the rsp and cdm probes.
5. Click OK, then click OK again to close the Raw Configure window.
   The dashboard_engine probe is automatically restarted if it was running when you opened the Raw Configure dialog. If the probe was not running, you must restart it manually.
View System Information in USM

To view performance and other information in the Unified Service Manager (USM) portlet, click **USM**.

**Note:** If the **USM** button is not activated (is grayed out), click the **Rebuild from template** icon (.REG).

If you linked to USM from a Dynamic View for a group, the **Members** table for the group is displayed in a new window. The **Members** table displays information about systems in the group. From here you can also access the **Alarms** tab, where you can view summary information about alarms for the group and manage alarms.

If you linked to USM from a Dynamic View for a device (a computer system, interface, or connectivity node), the **System** tab for that device is displayed in a new window. Depending on which probes are monitoring the system, the **System** tab displays information about system properties, alarms, disk usage, interface traffic, and performance. From here you can access other tabs where you can manage alarms or view metrics, group assignments, monitor assignments, and reports.

MiniMap Tool

The Mini Map tool allows you to adjust the zoom level and to pan around the dashboard. A small version of the dashboard is shown in the Mini Map window. Move the slider to adjust the zoom level. Drag the yellow field in the Mini Map to pan around the dashboard.
Viewing the Data as a Performance Report

You can view Dynamic View dashboard data in the format of a Performance Report.


The tables below the graphs contain the following information for the interval shown in the graph:

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Source</td>
<td>The computer hosting the probe sending the QoS data.</td>
</tr>
<tr>
<td>Target</td>
<td>The device/target being monitored, supplying the probe with QoS data.</td>
</tr>
<tr>
<td>Maximum</td>
<td>The maximum value for the interval shown.</td>
</tr>
<tr>
<td>Minimum</td>
<td>The minimum value for the interval shown.</td>
</tr>
<tr>
<td>Average</td>
<td>Average of the samples within the report period.</td>
</tr>
<tr>
<td>Stdev</td>
<td>Standard deviation of the samples within the report period.</td>
</tr>
</tbody>
</table>

As with any Performance Report, you can change the time interval for the report. Use the time selection buttons on the toolbar at the bottom of the report. Click the back ( youngster ) or forward ( punji ) icons to toggle between start/end date fields and quick view buttons.

Customizing the Dynamic View Templates

You can customize the Dynamic View report templates by editing them in the Dashboard Designer. The following sections contain information on template names, supported variables, and how to edit the templates.

Template Names

The following table lists the templates and which nodes in the Dynamic Views tree they are displayed for. In other words, in the Dynamic Views tree, when you click on a server that is monitored by the cdm probe, for example, the Dynamic View you see uses the _dtdv_server template.

<table>
<thead>
<tr>
<th>Template Name</th>
<th>Nodes Displayed For</th>
</tr>
</thead>
<tbody>
<tr>
<td>_dtdv_server</td>
<td>Server subnodes monitored by the cdm probe</td>
</tr>
<tr>
<td>Template Name</td>
<td>Nodes Displayed For</td>
</tr>
<tr>
<td>-----------------------</td>
<td>---------------------------------------------------------</td>
</tr>
<tr>
<td>_dtdv_server_rsp</td>
<td>Server subnodes monitored by the rsp probe</td>
</tr>
<tr>
<td>_dtdv_connectivity</td>
<td>Connectivity subnodes monitored by the net_connect probe</td>
</tr>
<tr>
<td>_dtdv_interface</td>
<td>Interface subnodes monitored by the interface_traffic probe</td>
</tr>
</tbody>
</table>
Supported Variables

When editing the Dynamic View templates, you can use a limited set of variables. Variables can be used in the following fields:

**Alarm widgets**

In the Set Alarm Filter dialog, accessed by clicking Select Alarm Filter at the bottom of the Alarm Properties pane, in the following fields:

- Host Name
- Subsystem
- Subsystem ID
- Domain
- Robot
- Probe
- Origin
- NAS

**Widgets that can connect to the QoS or NIS data sources**

- In the Target Index/Token field of the QoS tab of the Select Datasource dialog (not supported for connectivity templates)
- In the Query field of the NIS tab of the Select Datasource dialog (not supported for interface or connectivity templates)

**Text widgets**

The following lists the variables supported for each template.

**Server (_dtdv_server and dtdv_server_rsp templates)**

- $nimbus_address (not available for the dtdv_server_rsp template)
- $hubname (not available for the dtdv_server_rsp template)
- $has_qos
- $alarm_status
- $source_override
- $source
- $origin
- $domain
- $robotname
- $robotip
- $os_major
$os_minor
$os_version
$os_description
$os_user1
$os_user2
$cpu_usage
$memory_usage
$memory_paging

**Connectivity [**_dtdv_connectivity template]**

$host
$origin
$alarm_status
$icmp_response

**Interface Traffic [**_dtdv_interface template]**

$origin
$alarm_status
$device
$interface
$traffic
$errors
$discards
$in_utilization
$in_speed
$out_utilization
$out_speed

**More Information:**

[Using Variables for Widgets](#) (see page 143)
Edit a Dynamic View Template

You can edit the templates for Dynamic Views to customize them.

Follow these steps:

1. Open the Dashboard Designer.

2. Click the Import a dashboard from NimBUS or file icon in the Dashboard pane.
   The Import Dashboard From NimBUS dialog appears.

3. Click the name of the template to select it, then click Import Selected Dashboard. A notification dialog appears.

4. Click OK.

5. Double-click the template name in the Dashboard pane to open it.
   You can edit the template as any dashboard, except only certain variables are supported for alarm widgets.

Troubleshooting Dynamic Views

This section tells you how to troubleshoot common issues with Dynamic Views.

Dynamic Views May Display Redundant Tree Nodes

If you run the net_connect probe on two different robots to monitor the same systems, redundant tree nodes may appear in the left-hand navigation pane of the Dynamic Views portlet.

Note: Avoid running the net_connect probe from two different robots against the same systems.
Duplicate Data Series are generated

Symptom 1:
While using the CDM probe, dynamic views displays the updated data series when I switch from using "Short name for QoS Source" to "Long name QoS Source" or vice-versa only for the first time. When I switch from generating long or short names again, dynamic views does not display the updated data series.

Similarly, duplicate entries for the same node is available for selection while setting up a data source of the widget in dashboard designer.

Solution:
While using the CDM probe, if you switch between using Short name for QoS Source and Long name for QoS Source, two data series will be generated for the same QoS.

To see the updated data series in Dynamic Views as per the latest selection for generating QoS names, remove the latest entry manually from S_QOS_DATA table for the duplicated data series.

Symptom 2:
When CDM and RSP probes are simultaneously monitoring the same system, two data series for the same QoS are generated and only latest is shown in the dynamic views dashboard.

Solution:
Do not monitor any single system with both the CDM probe and the RSP probe simultaneously. Use either one of them.

Dynamic View is not Displayed

Symptom:
When I double-click on a server in the tree view of Dynamic Views, nothing happens and no dashboard is displayed in the right pane.

Solution:
This can be resolved by increasing the value for a timeout setting in the wasp probe.

1. In Infrastructure Manager, select the server in the tree view and double-click the wasp probe.
2. In the Raw Configure dialog, select the setup folder.
3. Double-click the nimpool_timeout key and change the value to 120.
4. Click OK.
5. Click Apply, and then close the window.
No Access Error

**Symptom:**

The Dynamic View dashboards do not load and I see an error that says "DashboardService SRV_ERR_NO_ACCESS".

**Solution:**

This may happen if you use the group_server probe to create groups and the **Name** and **Caption** fields have different values. To resolve this, enter the same values for the Name and Caption fields in the group_server probe. You can open the configuration GUI for group_server in Infrastructure Manager.

**Note:** Newer implementations of UMP use the nis_server probe instead of the group_server probe. To determine whether you are using group_server, check the `use_nis_server` setting in the `dashboard_engine.cfg` file. If it is set to 0, you are using group_server.

No Existing Interfaces Error

**Symptom:**

The Dynamic View dashboards do not load and I see a "none existing interfaces" error message in Dynamic Views.

**Solution:**

This may happen if all of these are true:

- You use the group_server probe. Newer implementations of UMP use the nis_server probe instead of the group_server probe. To determine whether you are using group_server, check the `use_nis_server` setting in the `dashboard_engine.cfg` file. If it is set to 0, you are using group_server.
- You have interfaces monitored by the interface_traffic probe that have a hyphen (-) in their names.
- You have enabled aggregated QoS measurements of interface traffic.

This occurs because the interface name is not recognized by the group_server probe. To resolve this, rename the interfaces (with no hyphens in the name) in the interface_traffic probe. You can open the configuration GUI for the interface_traffic probe in Infrastructure Manager.
Chapter 11: Flow Analysis

Chapter 12: List Designer

The List Designer allows administrators to design lists to be displayed in the List Viewer. The List Viewer displays information in a table format. The information can be in the form of text, numbers, gauges, alarms, or line graphs.

You can create two kinds of lists, detail or group. In detail lists, each row displays information for a single host or target. In group lists, each row displays aggregated information for a group of systems.

For group lists, you can specify a Drilldown Template (another list). Typically, the drilldown list contains data for the individual hosts or targets that make up the group. When viewing the group list in the List Viewer, you can click the name of a group to display the Drilldown Template.

For detail lists, you can click the name of a system to see a Drilldown Report with data for the system you clicked. The Drilldown Report is a Performance Report chart, created in Performance Reports Designer and specified in the Drilldown Report setting in the List Designer.

When you create a new list, by default it is a detail list. To create a group list, add groups to the list.

If you do not have the List Designer permission set in the Access Control List (ACL), you will see a "Permission Denied" message when you try to run the List Designer.

This section contains the following topics:
- Design a List (see page 198)
- User Tags (see page 200)
- List Designer Window (see page 201)
- Troubleshooting List Designer (see page 215)
Design a List

You can create a list to be displayed in the List Viewer application.

Follow these steps:

1. Click the New View icon.
2. From the Row Source pull down menu, select Hosts or Targets, then click Apply.
   Choose Hosts to display data for the systems doing the monitoring or Targets to display data for the endpoints of a test.
   For example, for disk usage choose Hosts to see information about the systems where CA Nimsoft Monitor (NM) is installed. For a URL test choose Targets to see information about the destination of the URL test.
3. Enter information in the remaining fields in the table definition area:
   - Column Header
     Allows you to give the source column a name other than Host or Target.
   - Hide Repeating Entries
     Displays host or target names only once in the source column in the List Viewer, making groups of entries easier to see.
   - Drilldown Report/Drilldown Template
     Specifies the item to display when you click the name of a group or host in the List Viewer to drill down for more information:
     - A list to be displayed when you click on the name of a group. The Drilldown Report can be any list, but typically it contains data for the individual systems that make up the group. This field name changes to Drilldown Template if the list has more than one group. The specified list is used as a template, displaying data for the members of whichever group is drilled down from.
     - A Performance Reports Designer chart to be displayed when you click on the name of a host or source. If no Performance Reports Designer chart is designated, the standalone USM is launched instead.
   Click the Find view icon ( ) next to the Drilldown Report field to browse to a list or chart to designate as the Drilldown Report or Drilldown Template.

   - Row Limit
     This sets the number of rows displayed in the list. For example, if you set this to 10, the top 10 rows for the sorting you have selected in the table are displayed.
     If you leave it set to 0, the List Viewer displays the number of rows that fit on a single page.
4. Add a group to the list if wanted:
   a. Enter information in the filter fields of the New Group pane.
   b. Click Apply.

5. Add more groups if wanted by clicking the Add group icon ( ) and then entering information in the fields in the group pane. The following fields are not self-explanatory:

   **Override Template**
   Allows you to drill down to a list other than the one specified for the Drilldown Template.

   **Static**
   Allows you to display data for the systems defined in the Override Template report instead of for the members of the group. This is most often used when working with virtual hosts.

   **Group name**
   Highlight the name of a group and enter a new name to rename it.

6. Add a column to the list by clicking Add Column.

7. Do the following:
   a. Select a QoS.
   b. If you are in Hosts mode (you selected Hosts from the Row Source pull down menu), select a target.

   The list of targets is populated once you select a QoS. If you are in Target mode the list of targets is not displayed since data will be displayed for all targets listed in the table. To restrict the targets listed in the table, add a group and use the filters in the New Group pane.

   c. If you are in Targets mode (you selected Targets from the Row Source pull down menu), click the Select All QoS check box if you want to display all QoS reported for the target in a single column. If you select this it is recommended that you also add a column with the Info data format (choose Info from the Display as pull down menu) and select QoS from the Info Type pull down menu. This allows you to see what each QoS is.

8. In the data format (right) pane, select the data format for the column from the Display as pull down menu.

9. Enter information for the column type you selected. For information about specific fields, see Add Column Panes (see page 205).

10. Click Apply.

11. Continue adding as many columns as you want.

   To change the order of the columns drag the column headers. To return to the group view, click the header of the source (host or target) column.
12. Click the Save icon.
   a. Select the directory location where you want to save the list and enter a name for the list. To save the list in a subdirectory, enter a directory structure such as servers/disk_usage/latest_list. This creates the subdirectories servers and disk_usage and the list named latest_list.
   b. Select **public**, **account**, or **private** from the drop-down list to the right of the name field.
   c. Click **Save**.

**User Tags**

You can filter on two user-defined criteria, called User Tags, in portlets that have filters. You can also automatically create groups according to User Tag.

**User Tag 1** and **User Tag 2** are text fields that are defined on the robot. You can enter any text in these fields. (Use Infrastructure Manager to open the configuration window for the robot.) Once you have done so, you can filter for these values in the Unified Service Manager, Performance Reports Designer, and List Designer portlets.

For example, you could use **User Tag 1** for geographic location and **User Tag 2** for type of server. For some systems you enter **Europe** for **User Tag 1** in the robot configuration, and for some systems you enter **Asia**. For **User Tag 2**, you enter either **production server** or **web server**.

You can then create reports for your regional systems by filtering on **User Tag 1** in the Performance Reports Designer (in the **Filters** tab) or List Designer (in the filter fields of the **New Group** pane). To do so, you would set the filter to **User Tag 1 is Europe** or **User Tag 1 is Asia**.
In the Unified Service Manager portlet, you can create separate groups for your production and web servers and apply different monitoring templates to them. To create the groups, in the Filters section of the Edit Group dialog you would set the filter to **User Tag 2 is production server** for one group, and **User Tag 2 is web server** for the other group.

**User Tag 1** and **User Tag 2** can be used to filter for any system that has a CA Nimsoft Monitor robot running on it. You cannot use **User Tag 1** and **User Tag 2** to filter for network devices that do not have a CA Nimsoft Monitor robot.

For information about automatically creating groups according to User Tag, see **Automatic Groups** (see page 447).

**List Designer Window**

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add Column button</td>
<td>Displays the <a href="#">Add Column panes</a> (see page 205), where you can define a column for a list.</td>
</tr>
<tr>
<td>Remove Column button</td>
<td>Removes the selected column from the table.</td>
</tr>
<tr>
<td>View as PDF icon</td>
<td>Opens the <a href="#">PDF Preferences Dialog</a> (see page 214), where you can choose settings for a PDF version of the list.</td>
</tr>
<tr>
<td>Refresh icon</td>
<td>Updates the data displayed in the table.</td>
</tr>
<tr>
<td>New view</td>
<td>Clears all fields or resets them to the defaults. Edit the settings to create a new list, then click the Save view ( ) or Save view as ( ) icon.</td>
</tr>
<tr>
<td>Open view icon</td>
<td>Opens the <a href="#">Open View</a> (see page 213) dialog, where you can select an existing list.</td>
</tr>
<tr>
<td>Save view as icon</td>
<td>Saves the list.</td>
</tr>
<tr>
<td>Feature</td>
<td>Description</td>
</tr>
<tr>
<td>-------------------------------</td>
<td>-----------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td><strong>Save view icon</strong></td>
<td>Opens the Save View (see page 214) dialog, where you can name the list and save it as either a public or private list. Public lists can be viewed by all users. Private lists can be viewed only by you. You can also use the Save View dialog to delete lists.</td>
</tr>
<tr>
<td><strong>Help icon</strong></td>
<td>Displays the online help for the List Designer.</td>
</tr>
<tr>
<td><strong>Table</strong></td>
<td>Displays the columns defined for the list. Drag the column headers to change the order of the columns. Enter text in a field next to a column header to filter for that text.</td>
</tr>
<tr>
<td><strong>Column Header field</strong></td>
<td>Enter a name for the column you are defining.</td>
</tr>
<tr>
<td><strong>Rows Source pull down menu</strong></td>
<td>Select whether the data is to be displayed for the hosts (the systems doing the monitoring) or the targets (the endpoints of a test). For example, for disk usage choose Hosts to see information about the systems where CA Nimsoft Monitor (NM) is installed. For a URL test choose Targets to see information about the destination of the URL test.</td>
</tr>
<tr>
<td><strong>Hide Repeating Entries check box</strong></td>
<td>Select this so that repeating names of hosts or targets are not displayed. For example, if there are multiple rows for the same host (source), the host is listed only once and that column is blank for subsequent rows for the same host. This makes it easier to see groups of entries.</td>
</tr>
<tr>
<td><strong>Apply button</strong></td>
<td>Updates the table with your changes.</td>
</tr>
<tr>
<td><strong>Drilldown Report/Template</strong></td>
<td>Specifies a list to use to drill down and view more information about the list. The name of this field changes from Drilldown Report to Drilldown Template for lists with more than one group. For a list that has only a single group, you can click the name of a system in the group in the List Viewer and drill down to a Performance Report for the system. The report is displayed with metrics for the system you clicked on. For a list with multiple groups, you can click the name of a group in the List Viewer to display the specified Drilldown Template. The Drilldown Template can be any list, but typically it contains data for the individual systems that make up the group. For example, you have groups named Data Center 1 and Data Center 2 made up of the systems at each data center. Your top-level list displays aggregated data for each data center. However, you want to drill down and see data for each system. To do this, you create a list named Data Center System Metrics with measurements for individual systems. You set the Data Center System Metrics template as the Drilldown Template. Now when you drill down from Data Center 1 or Data Center 2 aggregated data, you see the Data Center System Metrics report with data for the systems in that group. Enter the name of a list, or click the Find view icon (🔍) to the right of this field to select the list from a menu. The list specified as the Drilldown Template is used by default for all groups. To use a different drilldown template for a group, click the Override Template check box.</td>
</tr>
</tbody>
</table>
| **Edit button**  
(Displayed when a Drilldown Template is specified) | Allows you to edit the list specified as a Drilldown Template. You must save the current list in order to edit the Drilldown Template. This button is not available for lists with only a single group. In this case, the Drilldown Report is a Performance Report, which cannot be edited in the List Designer. |
<table>
<thead>
<tr>
<th>Feature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Row Limit</td>
<td>Limits the number of rows displayed. This makes loading the report in the List Viewer faster, especially if you have a large number of systems in the list. For example, if you set this to 10, the top 10 rows for the sorting you have selected in the table are displayed. If you sort the table by a CPU Usage column, the systems with the 10 largest CPU Usage values are displayed. If the Row Limit is set to 0, the report displays the number of reports that fit onto one screen. To see data for more systems, increase the Row Limit. <strong>Note:</strong> Setting the Row Limit to a very large number may make loading the list slow or cause the List Viewer to time out, depending on your environment. By default the List Viewer times out after 10 minutes.</td>
</tr>
<tr>
<td>Add group icon</td>
<td>Adds a New Group accordion, where you can create a group to display in the list. A group displays aggregated data for a set of hosts or targets, rather than data for each individual host or target.</td>
</tr>
<tr>
<td>Remove group icon</td>
<td>Deletes the current group.</td>
</tr>
<tr>
<td>Override Template check box</td>
<td>Select to specify a drilldown template for this group other than the one specified in the Drilldown Template field. Enter the name of a list, or click the Find view icon (🔍) to the right of this field to select the list from a menu.</td>
</tr>
<tr>
<td>Static check box</td>
<td>Select to display data for the systems defined in the Override Template report instead of for the members of the group. For example, if you have a group made up of VMware hosts you may want to see data not for the group members (hosts), but for the VMware instances on those hosts. To do so, use a report with the VMware instances defined as the data source as the Override Template and select the static check box.</td>
</tr>
</tbody>
</table>
**Edit button**

Allows you to edit the list specified as the Override Template. You must save the current list in order to edit the Override Template. This button is not available for lists with only a single group. In this case, the Drilldown Report is a Performance Report, which cannot be edited in the List Designer.

**Filter fields**

- Use these fields to display only specified hosts or targets in the list. For example, you can filter for only hosts monitoring a certain QoS measurement, or for all hosts or targets on a specified hub.
- not button - Click to display all items except the specified items.
- Source pull down menu - Select the item you want to filter for.
- Operator pull down menu - Select the operator (starts with, ends with, contains, or is) you want to use for the filter
- Text box - Enter the text you want to filter on.

**Add filter icon**

Click to add another row of filter options.

**Remove filter icon**

Click to remove the row of filter options. This icon is visible if you have more than one row of filter options.

### Add Column Panes

The Add Column panes are displayed when you click the Add Column button. The fields vary depending on whether you selected Hosts or Targets in the Row Source pull down menu.

**Hosts mode:**

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>QoS panel</td>
<td>Select the QoS measurement you want to display in the column.</td>
</tr>
</tbody>
</table>
**Target panel**

Select a target for the QoS measurement. This list is populated once you select a QoS measurement. If for the QoS measurement you selected the host name is part of the target, you see $HOST at the top of the host list. This variable allows you to create a single column for the data. Otherwise, because the host name is part of the target, you would have to create a new column for each system. Usually if the $HOST option is present you will want to use it.

**Data Format pane**

Use the fields in this pane to define the data to be displayed in the column. The fields and buttons in this pane change depending on the data format selected from the Display as pull down menu. For information on each field or button see the description for each data format:

- **Numeric** (see page 207)
- **Spark** (see page 207) (mini-graph)
- **Gauge** (see page 208)
- **Time Slice** (see page 209)
- **Alarm** (see page 211)
- **Status** (see page 212)
- **Info** (see page 213)

**Targets Mode:**

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Select All QoS check box</strong></td>
<td>Check to display all QoS reported for the target in a single column. If you select this it is recommended that you also add a column with the <strong>Info</strong> data format (choose <strong>Info</strong> from the <strong>Display as</strong> pull down menu) and select <strong>QoS</strong> from the <strong>Info Type</strong> pull down menu. This allows you to see what each QoS is. You can also use the filter fields in the <strong>New Group</strong> pane to restrict which QoS or targets are displayed (see List Designer Window for a description of these filter fields).</td>
</tr>
<tr>
<td><strong>QoS panel</strong></td>
<td>Select the QoS measurement you want to display in the column.</td>
</tr>
</tbody>
</table>
Data Format pane

Use the fields in this pane to define the data to be displayed in the column. The fields and buttons in this pane change depending on the data format selected from the Display as pull down menu. For information on each field or button see the description for each data format:

- **Numeric** (see page 207)
- **Spark** (see page 207) (mini-graph)
- **Gauge** (see page 208)
- **Time Slice** (see page 209)
- **Alarm** (see page 211)
- **Status** (see page 212)
- **Info** (see page 213)

### Numeric Data Format Pane

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Display as pull down menu</td>
<td>Select the data format for the column.</td>
</tr>
<tr>
<td>Apply button</td>
<td>Updates the table display with the changes you made.</td>
</tr>
<tr>
<td>Column Header field</td>
<td>Enter a name for the column.</td>
</tr>
<tr>
<td>Aggregation pull down menu</td>
<td>Choose the method you want to use to aggregate the data.</td>
</tr>
<tr>
<td>Time Period combo box and pull down menu</td>
<td>Select the number and units for the interval you want to see data for.</td>
</tr>
<tr>
<td>Show Units check box</td>
<td>Click to display the unit of the measurement, such as percent or second, in the column.</td>
</tr>
</tbody>
</table>

### Spark Data Format Pane

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Display as pull down menu</td>
<td>Select the data format for the column.</td>
</tr>
<tr>
<td>Apply button</td>
<td>Updates the table display with the changes you made.</td>
</tr>
<tr>
<td>Column Header field</td>
<td>Enter a name for the column.</td>
</tr>
</tbody>
</table>
Graph Style color box and pull down menu

Click the color box to open a palette where you can select a color for the graph items. Choose the type of graph, Line or Column.

Time Period combo box and pull down menu

Select the number and units for the interval you want to see data for.

Automatic Scale

Select to have the scale of the spark chart set automatically according to the minimum and maximum values for this column. Unselect this to set the minimum and maximum values for the spark charts in this column. This can be useful for seeing more differentiation in the specified range of values. For example, if the data for one system in the column has an outlier value it may tend to flatten the spark chart for the other columns. Excluding the outlier value from the minimum and maximum range has the effect of zooming in on the spark charts and displaying more differentiation within that range of values.

Graph Color

Select to open a color chart where you can set the color for the spark charts in this column.

Use Thresholds

Select to display colors in the spark charts that indicate thresholds. In the From and To fields, enter a range of values. Or, leave the From field blank to include all values up to the value in the To field. In the Color pull down menu, choose a color for the range of values. To add more rows, click the Add row icon (usahaan) to the right of the Color menu. To delete a row, click the Remove row icon (usahaan). Click the Move up or Move down arrows to change the order of the rows.

Gauge Data Format Pane

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Display as pull down menu</td>
<td>Select the data format for the column.</td>
</tr>
</tbody>
</table>
Apply button | Updates the table display with the changes you made.
---|---
Column Header field | Enter a name for the column.
Aggregation pull down menu | Choose the method you want to use to aggregate the data.
Time Period combo box and pull down menu | Select the number and units for the interval you want to see data for.
Show Value, Show Units | Whether to display numeric values on the gauge. If Show Value is selected, you can choose whether to display the units.
Max | Enter the maximum value to display.
Find Max Value icon | Finds the maximum value to date for the selected QoS measurement. Click the icon, then click Apply.
Thresholds fields | Enter threshold values for the color display in the gauge:
- All values up to Warn are green
- All values between Warn and Critical are yellow
- All values between Critical and Max are red

**Time Slice Data Format Pane**

The time slice data format is a bar graph where each bar represents aggregated data for an interval, or slice of time. You can define how long each time slice is. For example, if you want to see hourly data for a 24-hour period, set the Time Period to 24 Hours and the Slice Period to 1 Hour. You will see a bar graph with 24 bars, or time slices, where each slice represents an hour’s worth of data. You can set thresholds so that the color of each slice indicates the aggregated value of the data. You can also set a threshold so that if a specified number of null values is included in the data, the color of the slice indicates that.

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Display as pull down menu</td>
<td>Select the data format for the column.</td>
</tr>
<tr>
<td>Apply button</td>
<td>Updates the table display with the changes you made.</td>
</tr>
<tr>
<td>Column Header field</td>
<td>Enter a name for the column.</td>
</tr>
<tr>
<td><strong>Time Period spinner box and pull down menu</strong></td>
<td>Select the number and units for the interval you want to see data for.</td>
</tr>
<tr>
<td><strong>Slice Period spinner box and pull down menu</strong></td>
<td>Select the number and units for the length of each time slice (interval).</td>
</tr>
<tr>
<td><strong>Slice Aggregation pull down menu</strong></td>
<td>Choose the method you want to use to aggregate the data for each time slice.</td>
</tr>
<tr>
<td><strong>Detect Null Values check box</strong></td>
<td>Select if you want time slices with null values to be a different color. Some types of monitors return null values if no data is obtained. For example, when application response time is monitored, a null value is returned if the probe cannot contact the target web site. To see a visual indication that null values were returned, use this check box together with the Null Color selector and the Null Count Threshold. <strong>Note:</strong> In List Viewer you can double-click on a time slice to see a Performance Report chart of the data for that slice. Null values are represented in line graphs by a dotted line.</td>
</tr>
<tr>
<td><strong>Null Color selector</strong> <em>(Displayed only if Detect Null Values check box is checked)</em></td>
<td>Select the color for slices with null values. If the Null Count Threshold is reached, this color is displayed instead of the color defined in the Thresholds fields.</td>
</tr>
<tr>
<td><strong>Null Count Threshold</strong> <em>(Displayed only if Detect Null Values check box is checked)</em></td>
<td>Enter the number of null values that you want to trigger the change in color for a slice. For example, if you set the Null Color selector to blue and only want to see a slice be blue if it contains 3 or more null values, set the Null Count Threshold to 3. If a slice contains 2 null values it will be the color set in the Thresholds fields. If it contains 3 or more null values it will be blue. <strong>Note:</strong> This threshold is a count of the number of null values returned. This is different than the The Thresholds fields below, which reflect the aggregated value of the data.</td>
</tr>
</tbody>
</table>
Thresholds fields

Use these fields to assign a color for a specified range of values. These thresholds determine the color of each slice. In the From and To fields, enter a range of values for the color. Or, leave the From field blank to include all values up to the value in the To field. In the Color pull down menu, choose a color icon for the slice.

To add or delete rows, click the add row (+) or remove row (-) icons to the right of the Color pull down menu. You can also reorder the rows by clicking the Move up (↑) and Move down icons (↓).

Alarms Data Format Pane

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Display as pull down menu</td>
<td>Select the data format for the column.</td>
</tr>
<tr>
<td>Apply button</td>
<td>Updates the table display with the changes you made.</td>
</tr>
<tr>
<td>Status/Summary pull down menu</td>
<td>Whether to display alarm Status or Summary.</td>
</tr>
<tr>
<td>■ Status - Displays the highest severity of alarms. For example, if a system has three major alarms and one critical, critical is displayed as the status.</td>
<td></td>
</tr>
<tr>
<td>■ Summary - Displays a colored bar indicating the distribution of alarms. For example, if a system has three major alarms and one critical, the bar is 75 percent orange and 25 percent red.</td>
<td></td>
</tr>
<tr>
<td>Column Header field</td>
<td>Enter a name for the column.</td>
</tr>
<tr>
<td>Source Override field</td>
<td>Enter the name of a source (host or target) to see only alarms for that source.</td>
</tr>
<tr>
<td>Subsystem field</td>
<td>Enter the name of a subsystem to see only alarms for that subsystem.</td>
</tr>
<tr>
<td>Probe field</td>
<td>Enter the name of a probe to see only alarms for that probe.</td>
</tr>
<tr>
<td>User Tag field</td>
<td>Enter a User Tag to see only alarms with that User Tag.</td>
</tr>
</tbody>
</table>
# Status Data Format Pane

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Display as pull down menu</td>
<td>Select the data format for the column.</td>
</tr>
<tr>
<td>Apply button</td>
<td>Updates the table display with the changes you made.</td>
</tr>
<tr>
<td>Column Header field</td>
<td>Enter a name for the column.</td>
</tr>
<tr>
<td>Aggregation pull down menu</td>
<td>Choose the method you want to use to aggregate the data. If you choose Last, the last measurement is used and the time period is not relevant. If you choose a method other than Last, the Time Period fields are activated.</td>
</tr>
<tr>
<td>Time Period combo box and pull down menu</td>
<td>Select the number and units for the interval you want to see data for.</td>
</tr>
<tr>
<td>Thresholds fields</td>
<td>Use these fields to assign a status to a specified range of values. You can add more rows to create a series of status levels. In the From and To fields, enter a range of values for the status. Or, leave the From field blank to include all values up to the value in the To field. In the status icon pull down menu, choose a color icon for the status. In the Text field, enter the text to be displayed for data that falls in the specified range. For example, you could create a series of statuses that display the words low, medium, high, severe, and critical. To add more rows, click the add row icon (➕) to the right of the Text field.</td>
</tr>
</tbody>
</table>
Source Data Format Pane

The Info data format displays information about the system associated with the source, origin, or QoS.

Source column

This can be useful if you are viewing QoS measurements by target. For example, if the target you are monitoring is disk space on the C drive, the Source column tells you which system the C drive is on.

Origin column

Displays the origin for the source (host or target) for the row. Origins are a way of grouping data. For example, origins can represent data centers or MSP customers.

QoS column

The category of the data. This can be a useful column to display in target-based lists. Target names can be ambiguous and the QoS helps clarify this.

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Display as pull down menu</td>
<td>Select the data format for the column.</td>
</tr>
<tr>
<td>Apply button</td>
<td>Updates the table display with the changes you made.</td>
</tr>
<tr>
<td>Column Header field</td>
<td>Enter a name for the column.</td>
</tr>
<tr>
<td>Info Type</td>
<td>Type of data to display: Source, Origin, or QoS.</td>
</tr>
<tr>
<td>Hide Repeating Entries</td>
<td>Hides the name of repeating hosts or targets. For example, if there are multiple rows for the same host (source), the host is listed only once and that column is blank for subsequent rows for the same host. This makes it easier to see groups of entries.</td>
</tr>
</tbody>
</table>

Open View Dialog

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Open button</td>
<td>Displays the selected list in the List Designer.</td>
</tr>
<tr>
<td>Cancel button</td>
<td>Closes the dialog without displaying a list in the List Designer.</td>
</tr>
</tbody>
</table>
PDF Preferences Dialog

The PDF Preferences dialog allows you to choose settings for viewing a report as a PDF. The PDF can be printed or saved.

The PDF Preferences dialog has the following fields or buttons:

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Schedule Report icon (Not available from List Designer)</td>
<td>Opens the Create New Job dialog in the Report Scheduler portlet, where you can schedule the report to be run and delivered.</td>
</tr>
<tr>
<td>Orientation pull-down menu</td>
<td>Choose the page orientation for the PDF, Portrait or Landscape.</td>
</tr>
<tr>
<td>Size pull-down menu</td>
<td>Choose the page size for the PDF.</td>
</tr>
<tr>
<td>Quality pull-down menu</td>
<td>Choose the level of quality for the PDF.</td>
</tr>
<tr>
<td>OK button</td>
<td>Closes the dialog and displays the PDF.</td>
</tr>
<tr>
<td>Cancel button</td>
<td>Click to exit the dialog without creating a PDF.</td>
</tr>
</tbody>
</table>

Save Dialog

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Directory tree</td>
<td>Select a directory to save the list in.</td>
</tr>
<tr>
<td>Text field</td>
<td>Enter a name for the list. Enter a directory structure before the list name to create subdirectories. For example, servers/disk_usage/latest_list creates the subdirectories servers and disk_usage and the list named latest_list.</td>
</tr>
<tr>
<td>Access type pull down menu</td>
<td>Choose the type of access for the list:</td>
</tr>
<tr>
<td></td>
<td>■ public - Can be viewed by all users.</td>
</tr>
<tr>
<td></td>
<td>■ account - Can be viewed only by users of the account you are logged in as.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> Saving to an account requires the Portal Administrator ACL.</td>
</tr>
<tr>
<td></td>
<td>■ private - Can be viewed only by you.</td>
</tr>
<tr>
<td>Delete button</td>
<td>Deletes the list that is selected in the directory tree.</td>
</tr>
<tr>
<td>Save button</td>
<td>Saves the list.</td>
</tr>
</tbody>
</table>
**Cancel button**

Closes the dialog without saving the list.

---

**Disable Public Reports**

You can block users from saving Performance Reports and List Reports as public reports. You may want to do this if, for example, you are an MSP and you do not want your customers to be able to see reports from other UMP customers.

If saving public reports is disabled, users can save reports as either private or account reports. Users can still open public reports for viewing; they are only blocked from saving new reports as public reports.

**Follow these steps:**

1. Open Infrastructure Manager.
2. Click on the robot for the primary hub in the tree view.
3. Click the wasp probe to select it in the top right pane.
4. Press **CTRL** and right-click, then choose **Raw Configure** from the popup menu.
5. Click the **ump_common** folder in the tree view of the **Raw Configure** window.
6. Double-click **allow_public_reports** and change the value to **false**.
7. Click **OK**, then click **OK** again to close the **Raw Configure** window.
8. Restart the wasp probe.

---

**Troubleshooting List Designer**

This section tells you how to troubleshoot common issues with List Designer.
Time Slice Column Does Not Display Any Data

**Symptom:**
When I define a Time Slice column in List Designer, or look at a Time Slice column in List Viewer, the Time Slice column does not display any data.

**Solution:**
This occurs if the Time Slice column is too narrow for the slices to be at least one pixel wide.

Time slices may be narrower than a pixel if you enter a value in the Time Period field that is very long relative to the value you enter in the Slice Period field. For example, if you set the Time Period to 1000 Hours and the Slice Period to 1 Minute with the default column width, the slices are narrower than a pixel and are not displayed. Try decreasing the Time Period, increasing the Slice Period, or increasing the column width.

Time Slice Column Is Missing Data

**Symptom:**
When I define a Time Slice column in List Designer, or look at a Time Slice column in List Viewer, the Time Slice column is missing intervals of data in between time slices.

**Solution:**
The Time Slice column may appear to be missing data if the QoS sampling interval is longer than the Slice Period you selected. For example, if QoS sampling occurs every five minutes, and you select 1 Minute for the Slice Period field, the Time Slice column will display four minutes of blank space between each slice. To avoid this, use slice periods that are as long as or longer than the QoS sampling interval.
Chapter 13: List Viewer

The List Viewer displays information in a table format. The information can be in the form of text, numbers, gauges, alarms, or graphs.

Lists are configured in the List Designer application, or you can display predefined lists. The lists that you can view depend on your permissions.

If you do not have the List Viewer permission set in the Access Control List (ACL), you will see a "Permission Denied" message when you try to run the List Viewer application.

This section contains the following topics:

- Drilling Down in List Viewer (see page 217)
- List Viewer Window (see page 218)

Drilling Down in List Viewer

You can click in some cells in the List Viewer to drill down and see more information. The information displayed depends on the type of cell you click in.

**Group name**

Click on a group name in the source column to see a list with information about the members of the group. The list that is displayed is set in the List Designer, in the Drilldown Template or Override Template settings.

**Host or Source name**

Click on the name of a system in the Host or Source column to see a Performance Reports Designer chart with data for the system you clicked on. The report that is displayed is specified in the List Designer, in the Drilldown Report setting. If no Performance Reports Designer chart was specified as the Drilldown Report, the standalone USM is launched instead.

**Data cell**

Double-click on any data cell to see a Performance Reports Designer chart of that data.

**Note:** For a Time Slice column, double-clicking on a slice displays a Performance Reports Designer chart for the time slice, rather than the entire time period. However, when you hover your mouse over the column, a tool tip displays data for the entire time period.
Alarm cell

Double-click on a cell in an alarm column to open the Alarm Console. If you specified any values to filter for alarms (Source Override, Subsystem, Probe, or User Tag) when defining the column, you see only alarms that match those values.

More information:

- [Alarm Data Format Pane](#) (see page 211)
- [List Designer Window](#) (see page 201)

List Viewer Window

The List Viewer window contains the List Viewer table and a toolbar.

When working in the table, you can click on a column header to sort by that column. Click the header again to toggle the sort between ascending and descending order. Enter text in a field next to a column header to display only items containing that text. To change the order of the columns, drag the column headers.

You can also double-click in some cells in the table to drill down to more information. The information displayed depends on the type of cell you click in.

**Note:** For a Time Slice column, double-clicking on a slice displays a Performance Reports Designer chart for the time slice, rather than the entire time period. However, when you hover your mouse over the column, a tool tip displays data for the entire time period.

The List Viewer toolbar has these fields or buttons:

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>View as PDF icon</td>
<td>Displays the PDF Preferences dialog (see page 214), where you can choose settings for viewing the report as a PDF. The PDF can be printed or saved.</td>
</tr>
<tr>
<td>Refresh icon</td>
<td>Updates the data displayed in the List Viewer.</td>
</tr>
<tr>
<td>Open view icon</td>
<td>Opens the Open View (see page 219) dialog, where you can select a list for display in the List Viewer.</td>
</tr>
<tr>
<td>Help icon</td>
<td>Displays the online help for the List Viewer.</td>
</tr>
</tbody>
</table>
Preferences Dialog

Access the Preferences dialog by clicking the Settings icon (), then clicking Preferences.

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>List/Report/SLA Name field</td>
<td>Choose the name of the report you want to open by default in this instance of the portlet. If no report name is selected here, the portlet window contains no data when opened. After choosing a report and saving your preferences, log out and log back into UMP to see the specified report displayed by default.</td>
</tr>
<tr>
<td>Period field (Only for Performance Reports)</td>
<td>Choose the time interval to display for the default report.</td>
</tr>
<tr>
<td>Height field</td>
<td>Enter the height, in pixels, for this instance of the portlet to open to.</td>
</tr>
<tr>
<td>Maximized Height field</td>
<td>Enter the height, in pixels, for this instance of the portlet to display at when the maximize icon () is clicked.</td>
</tr>
<tr>
<td>Save button</td>
<td>Click to save your changes.</td>
</tr>
</tbody>
</table>

Open View Dialog

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Open button</td>
<td>Displays the selected list in the List Viewer.</td>
</tr>
<tr>
<td>Cancel button</td>
<td>Closes the dialog without displaying a list in the List Viewer.</td>
</tr>
</tbody>
</table>
Important! The Maintenance Mode portlet is deprecated. The Unified Service Manager portlet now provides maintenance mode functionality.

The Maintenance Mode application allows you to set the computer systems on your network to Maintenance Mode so they are temporarily unmonitored. The monitoring parameters for the system are retained, and when maintenance mode ends they are monitored again as before.

If you do not have the Maintenance Mode permission set in the ACL, you will see a "Permission Denied" message when you try to run the Maintenance Mode application.

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maintenance Mode icon</td>
<td>Click to display only systems in maintenance mode in the Host list.</td>
</tr>
<tr>
<td>Host filter field</td>
<td>Enter text to list only hosts with names that contain that text.</td>
</tr>
<tr>
<td>Status icons</td>
<td>Indicates the status of the host:</td>
</tr>
<tr>
<td>--------------</td>
<td>----------------------------------</td>
</tr>
<tr>
<td>■</td>
<td>= System is up</td>
</tr>
<tr>
<td>■</td>
<td>= System is down</td>
</tr>
<tr>
<td>(white)</td>
<td>= System is in maintenance mode</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Host list</th>
<th>Click a host name to select that host. You can Shift + click or Ctrl + click to select multiple hosts.</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Set button</th>
<th>Click to set the selected host(s) to maintenance mode. A calendar dialog opens, where you can set the time for maintenance mode to end.</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>End button</th>
<th>Click to end maintenance mode for the selected host(s). A confirmation dialog opens. If you end maintenance mode, the host is monitored again as it was before it was set to maintenance mode.</th>
</tr>
</thead>
</table>
Chapter 15: My Tickets

My Tickets offers a flexible display where you can view basic information about your CA Nimsoft Service Desk tickets. My Tickets displays a subset of the data available in the Service Desk and has a subset of the functionality. Unlike the Service Desk, My Tickets can be displayed on the same page as other portlets, allowing you to view your tickets alongside other key data for your organization.

The main view displays the Requests List, a table listing information about your tickets. From the Requests List you can click on a ticket to open it and view information about the ticket, add Work Log notes, or add attachments to the ticket.

This section contains the following topics:
- User Types (see page 223)
- Requests List (see page 224)
- Viewing Tickets (see page 224)
- Add a Note to a Ticket (Self Service Users) (see page 224)
- Add an Attachment to a Ticket (see page 225)
- Add A Work Log Item to a Ticket (Agent or Admin Users) (see page 225)

User Types

There are three types of users of Service Desk and My Tickets:

Self Service Users
Self service users are the end users who request IT support services. A self service user has limited interaction with Service Desk and My Tickets, using them mainly to submit requests and monitor the progress of their requests.

Service Desk Agents
Service Desk agents are the providers of IT support services. Service Desk agents, supervisors, and managers use Service Desk extensively to submit requests on behalf of end users across the organization and to track and monitor resolution and closure of requests.

Service Desk Application Administrators
Service Desk administrators are responsible for establishing the IT support process flow and setting up Service Desk to deliver IT support according to their organization’s requirements. Administrators set up Service Desk records and workflows and use Service Desk to ensure that IT support standards are met.
Requests List

The Requests List contains a table that displays information about tickets. The information displayed varies depending on the type of user you are. Self service users can toggle between viewing only the tickets they submitted and viewing all tickets for their organization. Agent and administrator users see tickets they submitted, tickets assigned to them, or tickets assigned to a group they are a member of.

Also, agent and administrator users see more columns in the Requests List table than self service users.

Click a column header to sort the table by that column.

Click on any row to open the ticket.

Viewing Tickets

You can open a ticket to view more information. Agent and administrator users see more information in the ticket than self service users.

From the ticket you can add a Work Log note or an attachment. To perform other actions, such as submitting new requests, performing searches, exporting data, printing tickets, or relating configuration items to the request, use the Service Desk.

To view a ticket

Click on any row in the Requests List to open a ticket.

Note: You can open only one ticket at a time. If you click on a row and the ticket is not displayed, check to see whether you already have a ticket window open.

Add a Note to a Ticket (Self Service Users)

Self service users can add a note to the Work Log for a ticket. Notes can be used to provide additional information about the issue or to communicate with the person who is working on the ticket.

To add a note to a ticket

1. Click on the ticket in the Requests list that you want to add the note to.
   The Modify Request window opens.
2. Scroll to the Work Log Information section.
3. Enter the note in the Description field.
4. Click Update.

The note is added to the ticket and is listed in the table in the Work Log Information section as a Client Note. Depending upon how Service Desk is configured at your company, the person assigned to work on your request may be notified that you have updated the request.

Add an Attachment to a Ticket

You can add an attachment to a ticket. This can be used to include images of error messages, data files, and so on. The maximum size for attachments is 3 MB. Once you add an attachment you cannot delete it.

To add an attachment to a ticket
1. Click on the ticket in the Requests list that you want to add the attachment to. The Modify Request window opens.
2. Scroll to the Attachments section.
3. Enter information in the Name and Description fields.
4. Click Browse and select the file you want to attach.
5. Click Upload.

The attachment is added to the ticket and is listed in the table in the Attachments section. You can view any attachments in the table by clicking on the row in the table.

Add A Work Log Item to a Ticket (Agent or Admin Users)

Agent or administrator users can add a work log item to a ticket.

To add a work log item to a ticket
1. Click on the ticket in the Requests list that you want to add the note to. The Modify Request window opens.
2. Scroll to the Activity Information section.
3. Enter information about the item in the Description field.
4. Select a Worklog Type from the drop-down menu.
5. Select Yes or No for the Client Viewable field depending on whether you want the client to be able to see the item.
6. Enter the number of minutes in the Time Spent field that you spent doing the item.
7. Click Update.

The item is added to the ticket and is listed in the table in the Activity Information section. If you selected Yes for the Client Viewable field, self service users will see the item in the Work Log Information table of the ticket.
Chapter 16: Nimsoft Remote Admin

The Nimsoft Remote Admin application provides a management console for discovery and configuration data in the CA Nimsoft Monitor database, Nimsoft Information Store (NIS).

**Note:** This portlet is replaced by the Discovery functionality in the Unified Service Manager portlet. However, this portlet remains available for legacy customers.

The Nimsoft Remote Admin portlet displays information about the computers in the network in a table format. It shows the state of all Configuration Items in the database. You can set the state for the computer systems discovered on the network.

Nimsoft Remote Admin supports multi-tenancy. Account and contact users see only discovered systems from the origin(s) assigned to the account they belong to. Accounts are created, and origins assigned, in the Account Admin portlet.

In order to access the Nimsoft Remote Admin application, you must have the Basic Management permission enabled in the Access Control List (ACL).

This section contains the following topics:

- Migration for Legacy Users (see page 228)
- Troubleshooting Nimsoft Remote Admin (see page 228)
Migration for Legacy Users

Unlike earlier versions of the Nimsoft Remote Admin product, formerly known as NIS Manager, the Nimsoft Remote Admin portlet does not manage monitoring. The Service Catalog functionality has been removed, and monitoring is now performed using the Unified Service Manager (USM) portlet.

If you previously used Nimsoft Remote Admin, your Service Catalogs are automatically migrated to USM in the following manner:

- USM groups are created for each Service Catalog under a container named Service Catalogs.
- USM monitoring templates are created and assigned to each Service Catalog.
- Service Elements are converted to USM monitors for the appropriate monitoring template. Monitors have the same name as the Service Elements.

Which devices are monitored depends on a setting you can choose while installing CA Nimsoft Monitor. If you select the unmanaged systems should be treated as managed option, all devices that have been discovered will be monitored.

Note: For implementations where discovery is already running, it is recommended that you do NOT select the unmanaged systems should be treated as managed option. Typically it is not desirable to monitor all previously discovered systems.

If this option is not selected, you must do the following in USM for devices that you want to monitor:

1. Set the device state to Managed. This can also be set in the Nimsoft Remote Admin portlet.
2. Make sure the devices are assigned to a USM group.
3. Make sure there is a monitoring template assigned to the USM group.

Note: For implementations where discovery has not run, it is recommended that you select the unmanaged systems should be treated as managed option. In this case you do not need to do step 1 above. All devices in a USM group that has a monitoring template assigned to it will be monitored.

Troubleshooting Nimsoft Remote Admin

This section tells you how to troubleshoot common issues with Nimsoft Remote Admin.
USM Templates Do Not Deploy

Symptom:
When I deploy USM templates in the Nimsoft Remote Admin portlet, I see the error message **Computer System State Error** in the **Status** column, and the templates do not deploy.

Solution:
This occurs if the **State** of the device is not set to **Managed**.

Do the following:
1. In Remote Admin, right-click the device in the tree pane and select **Set State**.
2. Under the **State** drop-down menu, select **Managed**.
Chapter 17: Performance Reports Designer

The Performance Reports Designer allows you to see a visual representation of QoS data. You select the host, QoS measurement, target, and time range, and the data is displayed as a chart. You can display multiple measurements on a single chart, and can view multiple charts at a time. You can drag charts between rows or drag a data series from one chart to another. You can choose the chart format (line, area, or column chart), and the Filters tab allows you to plot only the data that matches specified filters on the chart. You can easily change the source (host or target) of the data by using the Choose Source menu. You can save a set of charts as a report to print or to view later. The import and export features allow you to share charts with other users.

If you do not have the QoS Access permission set in the ACL, you will see a "Permission Denied" message when you try to run the Performance Reports Designer application.

This section contains the following topics:

- Understanding Performance Data (see page 231)
- Viewing Raw Performance Data (see page 232)
- QoS Selection Tabs (see page 233)
- Filters Tab (see page 234)
- User Tags (see page 235)
- Charts Pane (see page 236)
- Automatic Replacement of Fields in PRD Templates (see page 241)
- Toolbar (see page 242)
- Define Reports with a URL (see page 246)
- Troubleshooting PRD (see page 251)

Understanding Performance Data

Performance Reports Designer uses intervalized data. The software selects a data interval based on the scale of the chart you are viewing. The scale is affected by the timeframe you are viewing data for and the resolution of your screen. Performance Reports Designer selects the smallest interval so that there is only one data point per pixel. Possible intervals range from one minute to one day.

If there are multiple data samples for an interval, the values are averaged. For example, if the data is sampled every 5 minutes but the Performance Reports Designer chart is displaying 15-minute intervals, there are three data samples per interval. In this case, the average of the three data samples is displayed in the chart.
Conversely, if there are fewer data samples than intervals, Performance Reports Designer displays interpolated data. For example, if the data is sampled every 15 minutes but Performance Reports Designer is displaying 5-minute intervals, two of the three data points for each interval are interpolated. If the data is interpolated, when you mouse over the data point in the chart the data tip labels the number as Estimate rather than Value.

Performance Reports Designer also interpolates data for periods where data is missing. Missing data is indicated by a dotted line in a line or area chart. In a bar chart, missing data is interpolated but there is no visual indication. However, if you stack multiple data series on a bar chart and one data series has fewer data samples, you see shadow columns for interpolated data. This allows you to stack a denser data series on top of a sparser one.

For bar charts, each column displays data for five times the interval for the chart. For example, if the chart is using 1-minute intervals, each column represents 5 minutes.

If you export the Performance Reports data to a CSV file, only the actual data samples are included; interpolated data is not included in the CSV file.

**Viewing Raw Performance Data**

You can view the raw data collected for a chart, rather than the intervalized data. You may want to do this if you want to see the actual maximum and minimum values, or if you want to see details such as spikes in the data. However, because it may require retrieving and displaying a large amount of data, performance of the portlet may be slower.

To view raw data for a chart, click the Configure icon (🔗) and choose Raw Data from the menu. Raw data is displayed in the chart, and the aggregates listed below the chart (Min, Max, and Standard Deviation) also reflect the raw data.

The raw data setting is saved as a property of the chart. If you refresh your browser or reopen the chart, it continues to display raw data. If you add a data series, it will also display raw data.

To return to viewing intervalized data, click the Configure icon (🔗) and choose Raw Data again to de-select it.

You cannot display raw data for stacked charts, and you cannot stack raw data charts. Due to performance issues, it is not recommended to display raw data in column style charts.
QoS Selection Tabs

The QoS Selection tabs are where you can select the host, QoS measurement, and target you want to view data for. There are two QoS Selection tabs, the QoS Selection by Host tab and the QoS Selection by Target tab. The two tabs function the same way, but the order in which you select hosts and targets is different.

- QoS Selection by Host - Use this to easily find multiple measurements from the same host
- QoS Selection by Target - Use this to easily find measurements of the same type on multiple hosts

The selection panes have the following fields:

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Host</td>
<td>Host name of the computer you want to chart measurements for.</td>
</tr>
<tr>
<td>QoS</td>
<td>Type of QoS measurement you want to chart.</td>
</tr>
<tr>
<td>Target</td>
<td>Which measurement, if more than one measurement for the selected QoS type is taken. For example, for CPU usage you can choose Idle, System, User, Wait, or &lt;host name&gt;.</td>
</tr>
</tbody>
</table>

Work from left to right in the selection panes. Enter text in the filter field at the top of each selection pane to display only items that contain that text.

To select a measurement for charting in the QoS Selection by Host tab, for example:

1. In the Host pane, select the host name of the computer you want to chart measurements for.
   
   Once you select the host, the QoS measurements for tests you have running on that host are listed in the QoS pane.

2. In the QoS pane, select a QoS measurement.
   
   Once you select a QoS measurement, the available targets are listed in the Target pane.

3. In the Target pane, select a target, then drag it to a chart pane to display it.

   You can drag multiple targets to each chart. If the targets are measured in different units, a second y-axis is added on the right side of the chart. You can add as many targets with the units on the two y-axes as you want, but you cannot add a target with different measurement units. For example, if the left y-axis represents percent and the right y-axis represents seconds, you can add an unlimited number of targets measured in percent or seconds, but you cannot add a target that is measured in megabytes.
## Filters Tab

The Filters tab is where you can create filters to narrow down the data to be charted. Once you enter your filter options, check the results with the Check Filters button to make sure the results are not too broad, then drag the drag bars to a chart. Charts update dynamically at a regular interval to reflect changes in the search results.

The Filters tab has the following fields:

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name field</td>
<td>Enter a name for the filter.</td>
</tr>
<tr>
<td>Check Filters button</td>
<td>Displays the items that match the filter in the pane to the right. Use this to make sure the filter is narrow enough that you do not bog down your system charting a huge amount of data.</td>
</tr>
<tr>
<td>Apply button</td>
<td>Applies changes to the filter to the currently displayed charts.</td>
</tr>
<tr>
<td>Drag bars</td>
<td>Drag these bars to a chart to graph the results for the filter.</td>
</tr>
<tr>
<td>Add filter icon</td>
<td>Click to create a new filter.</td>
</tr>
<tr>
<td>Remove filter icon</td>
<td>Deletes the current filter. Visible when you have multiple filters.</td>
</tr>
<tr>
<td>Filter options</td>
<td>Use these fields to display only data from specified hosts or targets in the chart. For example, you can filter for only hosts monitoring a certain QoS measurement, or for all hosts or targets on a specified hub.</td>
</tr>
<tr>
<td></td>
<td>(blank)/not button - Click to display all items except the specified items.</td>
</tr>
<tr>
<td></td>
<td>Criterion pull-down menu - Select the item you want to filter for, such as Source, Probe, QoS, and so on.</td>
</tr>
<tr>
<td></td>
<td>Operator pull-down menu - Choose the appropriate operator, such as is, contains, starts with, and so on.</td>
</tr>
<tr>
<td></td>
<td>Text field - Enter the appropriate text for the criterion you chose.</td>
</tr>
<tr>
<td>Add filter row icon</td>
<td>Click to add another row of filter options.</td>
</tr>
<tr>
<td>Remove filter row icon</td>
<td>Click to remove the row of filter options. This icon is visible if you have more than one row of filter options.</td>
</tr>
</tbody>
</table>
Up/down icons

Click to move the row up or down. Filter rows are applied in sequential order. These icons are visible if you have more than one row of filter options.

**Note:** The special characters \, %, and _ may need to be preceded by or enclosed in escape characters when you use basic search or advanced search, or when you apply filters in USM or in PRD. In the case of advanced search and filters, this applies when the contains, starts with, or ends with operator is selected. The escape character required depends on the type of database. For MySQL, the backslash (\) is the escape character; for MS SQL, special characters are enclosed in square brackets ([ ]). Searching for special characters is not supported for Oracle.

The following table shows how to enter special characters for each database type.

<table>
<thead>
<tr>
<th>Character</th>
<th>MySQL</th>
<th>Oracle</th>
<th>MS SQL</th>
</tr>
</thead>
<tbody>
<tr>
<td>\</td>
<td>\</td>
<td>Not applicable</td>
<td>\ (No escape character required)</td>
</tr>
<tr>
<td>%</td>
<td>%</td>
<td>Not applicable</td>
<td>[%]</td>
</tr>
<tr>
<td>_ (underscore)</td>
<td>_</td>
<td>Not applicable</td>
<td>[ ]</td>
</tr>
</tbody>
</table>

### User Tags

You can filter on two user-defined criteria, called User Tags, in portlets that have filters. You can also automatically create groups according to User Tag.

**User Tag 1** and **User Tag 2** are text fields that are defined on the robot. You can enter any text in these fields. (Use Infrastructure Manager to open the configuration window for the robot.) Once you have done so, you can filter for these values in the Unified Service Manager, Performance Reports Designer, and List Designer portlets.

For example, you could use **User Tag 1** for geographic location and **User Tag 2** for type of server. For some systems you enter Europe for **User Tag 1** in the robot configuration, and for some systems you enter Asia. For **User Tag 2**, you enter either production server or web server.
You can then create reports for your regional systems by filtering on **User Tag 1** in the Performance Reports Designer (in the Filters tab) or List Designer (in the filter fields of the New Group pane). To do so, you would set the filter to **User Tag 1 is Europe** or **User Tag 1 is Asia**.

In the Unified Service Manager portlet, you can create separate groups for your production and web servers and apply different monitoring templates to them. To create the groups, in the Filters section of the Edit Group dialog you would set the filter to **User Tag 2 is production server** for one group, and **User Tag 2 is web server** for the other group.

**User Tag 1** and **User Tag 2** can be used to filter for any system that has a CA Nimsoft Monitor robot running on it. You cannot use **User Tag 1** and **User Tag 2** to filter for network devices that do not have a CA Nimsoft Monitor robot.

For information about automatically creating groups according to User Tag, see [Automatic Groups](#) (see page 447).

**Charts Pane**

The charts pane is where you can view the data for the selected host, QoS, and target. You can select the chart format, maximize the display, or adjust the data range for the y-axis. Hover over a data point to display information about the data in a pop-up window. You can drag charted data to move it from one chart to another. For a line chart, for example, click on a line and drag it to another chart.

To zoom in on an area of a chart, drag the cursor to select that area. Repeat to zoom in again. Click the graph to zoom all the way out. Drag the cursor to select a portion of an axis to zoom in on the selected range.
The charts have the following fields:

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Title bar</td>
<td>Double-click on Title to enter a title for the report.</td>
</tr>
<tr>
<td>Style icon</td>
<td>Click on a data series in the chart, and then click the <strong>Style</strong> icon to apply one the following styles, or to remove a data series from the chart:</td>
</tr>
<tr>
<td></td>
<td>■  = Line chart</td>
</tr>
<tr>
<td></td>
<td>■  = Area chart</td>
</tr>
<tr>
<td></td>
<td>■  = Column chart</td>
</tr>
<tr>
<td></td>
<td>■  = Removes the selected data series from the chart</td>
</tr>
<tr>
<td>Choose Source combo box</td>
<td>Click to chart the same set of data series for another host or target.</td>
</tr>
<tr>
<td></td>
<td>If you have selected multiple targets for a single host (QoS Selection by Host), clicking this displays a list of hosts with the same QoS measurements and targets. This allows you to easily view the same set of measurements for different hosts.</td>
</tr>
<tr>
<td></td>
<td>If you have selected a single target for multiple hosts (QoS Selection by Target), clicking this displays a list of targets for the selected QoS measurement. This allows you to easily view different measurements for the same hosts.</td>
</tr>
<tr>
<td></td>
<td>The menu bar displays the name of the current host or target. If there are no other corresponding hosts or targets, the combo box is not displayed.</td>
</tr>
<tr>
<td>Configure icon</td>
<td>Click to display a pop-up menu with the options below.</td>
</tr>
<tr>
<td>----------------</td>
<td>--------------------------------------------------------</td>
</tr>
<tr>
<td></td>
<td>■ Stacked - Stacks the data series on top of each other.</td>
</tr>
<tr>
<td></td>
<td>■ Legend - Toggles the display of the legend on and off. The legend, which appears at the bottom left of the chart, displays the host/QoS/target for each data series in the chart.</td>
</tr>
<tr>
<td></td>
<td>■ Raw Data - Displays all data collected, rather than intervalized data. This allows a detailed view of the data and displays the actual minimums and maximums, but does result in slower performance.</td>
</tr>
<tr>
<td></td>
<td>■ Title - Adds a title to the chart. Double-click the word Title at the top of the chart, then enter your text.</td>
</tr>
<tr>
<td></td>
<td>■ Info text - Adds descriptive information to the bottom of the chart. Double-click the words Info text, then enter your text.</td>
</tr>
<tr>
<td></td>
<td>■ Trend Line - Displays a trend line for the data. If you hover over a data point on the chart, the trend value for the data point is included in the tooltip.</td>
</tr>
<tr>
<td></td>
<td>■ Baseline - Displays baseline data, if available, as a dotted stair-step line. If you hover over a data point on the chart, the baseline value for the data point is included in the tooltip.</td>
</tr>
<tr>
<td></td>
<td>■ Percentile - Adds a line marking a specified percentile to the chart. Enter the percentile in the spinner box.</td>
</tr>
<tr>
<td></td>
<td>■ Scale - Adds a scale factor to the selected data series. For example, a scale of 2 doubles the values of the series. Once you choose Scale from the pop-up menu, a text box is displayed on the title bar where you can enter a number for the scale factor you want.</td>
</tr>
</tbody>
</table>

<p>| Maximize icon | Maximizes the display of the chart within the portlet window. |</p>
<table>
<thead>
<tr>
<th><strong>Charts Pane</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>View CSV data</strong></td>
</tr>
<tr>
<td><strong>Generate chart URL</strong></td>
</tr>
<tr>
<td><strong>Clear icon</strong></td>
</tr>
<tr>
<td><strong>Remove chart icon</strong></td>
</tr>
<tr>
<td><strong>Data range buttons</strong></td>
</tr>
<tr>
<td><strong>Page up icon</strong></td>
</tr>
<tr>
<td>Move row up/Move row to previous page icon</td>
</tr>
<tr>
<td>Add row icon</td>
</tr>
<tr>
<td>Add chart icon</td>
</tr>
<tr>
<td>Move row down/Move row to next page icon</td>
</tr>
<tr>
<td>Page down icon</td>
</tr>
</tbody>
</table>
Automatic Replacement of Fields in PRD Templates

You can enter parameterized syntax, or variables in the title bar, chart title, or in the chart info text of a PRD template. When you save the template and then view the report for a system, the parameterized values are automatically replaced with actual values for the system.

For example, if you enter $name: $ip in the title bar of a template, when you view the report for a system the system name and IP address are displayed as the title of the report.

You can use the following variables:

- $caption
- $description
- $name
- $domain
- $origin
- $ip
- $dns_name
- $mac
- $oui_organization
- $os_type
- $os_name
- $os_version
- $os_description
- $dedicated
- $ad_name
- $ad_domain
- $nb_name
- $nb_domain
- $user_tag_1
- $user_tag_2
The toolbar allows you to select the time range to display data for, view or print a report as a PDF, save a report, or view previously saved reports.

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Undo/Redo icons</td>
<td>Click to undo a change you made, or to redo it after undoing it.</td>
</tr>
<tr>
<td>Time selection fields:</td>
<td></td>
</tr>
</tbody>
</table>
Click the back (◀) or forward (▶) icons to scroll through the time selection options:

- **Start/end date fields** - Useful for charting a specific time period. Click to display a calendar, where you can select the start or end time for the data you want to chart. If the end time is the current time or is in the future, the word **Now** is displayed in the date box and the charts are updated at 1-minute intervals.

- **Date and time sliders** - Useful for browsing the same time period on different days. To use the sliders, work from left to right. The lefthand slider lets you select the level of granularity you want (year, month, day, or hour). Once you select that, the middle slider lets you select which year, month, day, or hour you want to chart. Once that is selected, the righthand slider lets you select a period within that year, month, day, or hour.
  
  For example, if you are viewing data for 3 to 5 p.m. for the current day and want to view data for the same time the previous day, move the middle slider to the left one increment. (If you are viewing data for a specific day, the lefthand slider is already set to Day.)
  
  You cannot move the sliders beyond the increment for the current time.

- **Quick View buttons** - Useful for viewing data for common time intervals:
  
  - [◀] = Previous hour
  - [1] = Previous day
  - [7] = Previous week
  - [30] = Previous month

  The interval is a sliding window so that as time passes new values are appended and old values are dropped.
PDF Preferences Dialog

The PDF Preferences dialog allows you to choose settings for viewing a report as a PDF. The PDF can be printed or saved.

The PDF Preferences dialog has the following fields or buttons:

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Schedule Report icon (Not available from List Designer)</td>
<td>Opens the Create New Job dialog in the Report Scheduler portlet, where you can schedule the report to be run and delivered.</td>
</tr>
<tr>
<td>Orientation pull-down menu</td>
<td>Choose the page orientation for the PDF, Portrait or Landscape.</td>
</tr>
<tr>
<td>Size pull-down menu</td>
<td>Choose the page size for the PDF.</td>
</tr>
<tr>
<td>Quality pull-down menu</td>
<td>Choose the level of quality for the PDF.</td>
</tr>
<tr>
<td>OK button</td>
<td>Closes the dialog and displays the PDF.</td>
</tr>
<tr>
<td>Cancel button</td>
<td>Click to exit the dialog without creating a PDF.</td>
</tr>
</tbody>
</table>

Save Dialog

The Manage Reports dialog allows you to save a set of charts as a report, open or delete previously saved reports, and import or export reports.

The Manage Reports dialog has the following fields and buttons:

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reports tree</td>
<td>Click the name of a previously saved report to select it.</td>
</tr>
</tbody>
</table>
Reports text box Displays the name of a report selected in the Reports tree. Or, enter the name of a new report.

Account type pull-down menu Choose the type of account access for the report. If you are an administrator, you can assign the report to a specific account.
- public - Users from all accounts can view the report.
- account - Can be viewed only by users of the account you are logged in as.
Note: Saving to an account requires the Portal Administrator ACL.
- private - Only you can view the report.

Delete button Deletes the selected report.
Import button Imports a previously exported report.
Export button Exports the selected report to an XML file.
Open button Opens the selected report.
Save button Saves the report named in the Reports text box.
Cancel button Closes the dialog without taking further action.

Disable Public Reports

You can block users from saving Performance Reports and List Reports as public reports. You may want to do this if, for example, you are an MSP and you do not want your customers to be able to see reports from other UMP customers.

If saving public reports is disabled, users can save reports as either private or account reports. Users can still open public reports for viewing; they are only blocked from saving new reports as public reports.

Follow these steps:
1. Open Infrastructure Manager.
2. Click on the robot for the primary hub in the tree view.
3. Click the wasp probe to select it in the top right pane.
4. Press CTRL and right-click, then choose Raw Configure from the popup menu.
5. Click the ump_common folder in the tree view of the Raw Configure window.
6. Double-click allow_public_reports and change the value to false.
7. Click OK, then click OK again to close the Raw Configure window.
8. Restart the wasp probe.

Preferences Dialog

Access the Preferences dialog by clicking the Settings icon ( ), then clicking Preferences.

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>List/Report/SLA Name field</td>
<td>Choose the name of the report you want to open by default in this instance of the portlet. If no report name is selected here, the portlet window contains no data when opened. After choosing a report and saving your preferences, log out and log back into UMP to see the specified report displayed by default.</td>
</tr>
<tr>
<td>Period field (Only for Performance Reports)</td>
<td>Choose the time interval to display for the default report.</td>
</tr>
<tr>
<td>Height field</td>
<td>Enter the height, in pixels, for this instance of the portlet to open to.</td>
</tr>
<tr>
<td>Maximized Height field</td>
<td>Enter the height, in pixels, for this instance of the portlet to display at when the maximize icon ( ) is clicked.</td>
</tr>
<tr>
<td>Save button</td>
<td>Click to save your changes.</td>
</tr>
</tbody>
</table>

Define Reports with a URL

You can use URLs with parameters to define reports from a browser without being logged into UMP. You may want to do this if you want to share reports with people who do not have UMP accounts, or to link to reports from a third-party application.

You can view reports as defined, and you can change parameters such as the source, QoS, target, or time period by specifying one in the URL. For example, if you have a report named CPU that displays a series of CPU measurements, you can view the same measurements for a different computer system by specifying a different source in the URL.

You can define one or more data series for each chart, and you can have multiple charts in a row and multiple rows.
The syntax for the URL is:

http://<ump_server>/qoschart/jsp/standalone.jsp?user=<user>&password=<pass>&def=<def>

If you omit the user and password the user will be prompted for credentials.

**The def parameter**

The def parameter uses the JSON syntax, with single quotes rather than double quotes. The full specification can be viewed at [http://www.json.org/](http://www.json.org/).

The JSON syntax is a set of key value pairs specified as `{key: value, key2: value2}`.

All of the options except time specifications are listed in an array to define sets of charts. Each set of charts is listed in an array to define a set of rows.
Define Reports with a URL

Example - Multiple Rows

Note: Normally the URL is specified in a single line. However, to make it easier to read these examples are broken into multiple lines.

def=
    {'time': '-1hour', 'rows': [
        {'charts': [
            {'parms': '...'},
            {'series': [
                {'sqt': 'myserver|QOS_CPU_USAGE|Total', 'scale': '1', 'color': '8000', 'style': 'line'},
                {'sqt': 'myserver|QOS_CPU_USAGE|User', 'scale': '1', 'color': '800000', 'style': 'line'}
            ]},
            {'parms': '...'},
            {'series': [
                {'sqt': 'myserver|QOS_CPU_USAGE|Total', 'scale': '1', 'color': '8000', 'style': 'line'},
                {'sqt': 'myserver|QOS_CPU_USAGE|User', 'scale': '1', 'color': '800000', 'style': 'line'}
            ]},
            {'charts': [
                {'parms': '...'},
                {'series': [
                    {'sqt': 'myserver|QOS_CPU_USAGE|Total', 'scale': '1', 'color': '8000', 'style': 'line'},
                    {'sqt': 'myserver|QOS_CPU_USAGE|User', 'scale': '1', 'color': '800000', 'style': 'line'}
                ]},
                {'parms': '...'},
                {'series': [
                    {'sqt': 'myserver|QOS_CPU_USAGE|Total', 'scale': '1', 'color': '8000', 'style': 'line'},
                    {'sqt': 'myserver|QOS_CPU_USAGE|User', 'scale': '1', 'color': '800000', 'style': 'line'}
                ]}
            ]}
        ]}
    ]

Example - Single Row

To define a single row you can omit the 'row' option.

def=
    {'time': '-1hour', 'charts': [
        {'parms': '...'},
        {'series': [
            {'sqt': 'myserver|QOS_CPU_USAGE|Total', 'scale': '1', 'color': '8000', 'style': 'line'},
            {'sqt': 'myserver|QOS_CPU_USAGE|User', 'scale': '1', 'color': '800000', 'style': 'line'}
        ]}
    ]
Similarly, you can omit the charts parameter if you only have one chart.

Syntax

Strings are enclosed in single quotes.

Because this is a URL, certain characters, including single quotes, must be properly escaped:

- Single quotes are written as `\'` or `\`
- The ampersand (`&`) is written as `_amp_
- The backslash is written as `\\`

Numbers are written as numbers.

An array is a set of JSON objects enclosed in brackets.

For `<boolean>` `y`, `yes`, `1`, `t`, `true`, or `on` can be used for true. For false you can use `n`, `no`, `0`, `f`, `false`, `false`, or `off`.

The time frame must be specified at the top level of a report definition:

`'time'`

Time parameters for the report.

- `<epoch>` is a specific time in seconds since epoch
- `<period>` is a time period supporting the units "sec", "min", "hour", "day", "month", "year". For example, "60min" or "2hour"

Time can be specified as follows:

- `'<epoch>-<epoch>'` - start time to end time
- `'<epoch>-'` - start time to now
- `'<epoch><period>` - the `<period>` leading up to `<epoch>`
- `'<epoch>+<period>` - the `<period>` after `<epoch>`
- `'<period>` - the last `<period>`. For example, `'60min` shows the last 60 minutes.

You can set a flag for getting the raw data:
'rawData'
Use with a value of true to see the data in CSV format instead of a graph or chart.

A report consists of some number of rows of charts. If a single row of charts is desired, the 'rows' parameter may be omitted:
'rows'
An array of row definitions. A row contains one or more charts.
'charts'
An array of chart definitions. A chart contains one or more series.

The parameters for a chart are:
'title'
Title of the report. If omitted there is no title.
'info'
Informational text displayed at the bottom of the chart.
'tref'
Time reference. If a time is specified, you can highlight a time or a time period.
   '<epoch>+-<period>,[<epoch>+-<period>]'- Start and stop of the highlighted period (stop is optional).
   "S" for epoch is the chart start time. "E" is the end time.
   Example: 'tref':'S+10min,E-1hour' highlights the area from 10 minutes after the chart starts to 1 hour before it ends.
'stacked'
<boolean> Stacks the series for the chart.
'lmax'
Sets the maximum value for the left axis.
'lmin'
Sets the minimum value for the left axis.
'rmax'
Sets the maximum value for the right axis.
'rmin'
Sets the minimum value for the right axis.
'legend'

<boolean> to turn it off. Use an empty string to only show labels. Specific legend annotations can be specified and ordered.

'M' - maximum value
'm' - minimum value
'l' - last value
't' - total of all values
'a' - average value
'p' - show the percentile if it's turned on for the series

Example: 'legend':'Mpla' would show the max, percentile, last value, then average.

'series'

An array of series definitions.

Series are defined as follows:

'sqt'

Specifies the data for the series as '<source>|<qos>|<target>'. An asterisk (*) can be used as a wild card for a full or partial word.

'scale'

Scales the series by this factor.

'ptile'

Shows the specified percentile, for example 'ptile':95 shows the 95th percentile for the series.

'style'

Series style:

- 'line' - line chart
- 'area' - area chart
- 'col' - column chart

'color'

Color of the series specified in RGB in hex. For example, FF0000 is red.

Troubleshooting PRD

This section provides troubleshooting information for PRD.
Performance Reports May Convert Incorrectly to PDF

Symptom:

When I convert a PRD report containing many charts to a PDF, the PDF version does not display correctly.

Solution:

When you convert a PRD report with a large number of charts to a PDF, the system tries to place the entire report, regardless of the number of charts it contains, on one page of PDF output.

For PRD reports to display correctly in PDF format, it is recommended that you include no more than four or five charts per PRD report. If your PRD report contains more than four or five charts, remove charts as needed until the PDF output displays correctly. Depending on the number of charts in your report, you may be able to correct this issue in printed PDF output by selecting a larger paper size.
Chapter 18: Relationship Viewer

This section contains the following topics:

- **Overview** (see page 253)
- **Using the Relationship Viewer** (see page 255)
- **Launching the Relationship Viewer using the WebContent Portlet** (see page 265)

**Overview**

This section provides conceptual information about the Relationship Viewer.

**About the Relationship Viewer**

The Relationship Viewer portlet provides a graphical view of managed elements and their relationships to one another. Managed elements appear as icons, connected by lines that represent the relationships between them.

The Relationship Viewer displays information that is created by various applications—you might have any number of them. With this flexibility, the Relationship Viewer can depict almost any kind of objects and their relationships:

- Computers and network device icons, with the relationship line indicating a network connection between two of them.
- User and service icons, with the relationship line indicating a user’s subscription to a service.
- Physical locations, with the relationship line indicating perhaps a direct transportation route or some other link between locations.

Remember, the underlying applications determine what objects and relationships are available for presentation in the Relationship Viewer.

In addition, the icons and lines may each have visual attributes that depict the state of the element or relationship. For example, if the underlying data provides it, the operational status of a router may be shown as in a "Normal" state, "Failure" state, or other states.

Controls in the Relationship Viewer let you navigate the view in various ways, such as zooming, panning, altering the layout, and so on.
Clicking the mouse on an icon or connecting line causes a descriptive panel to pop-up. For example, clicking on a router icon will display a panel that includes key information about the router including its name, state, address, and so on. Some descriptive panels may offer buttons or links to further detail about the object.

More information:

Accessing the Relationship Viewer (see page 254)
Using the Relationship Viewer (see page 255)

Accessing the Relationship Viewer

There are three ways to access the Relationship Viewer:

Portal access via the USM Actions menu

The most common way to access the Relationship Viewer is by choosing Launch Relationship Viewer from the Actions menu in the Unified Service Manager (USM) portlet.

Note: The Launch Relationship Viewer menu option is present only if the relationship_services probe is running and at least one relationship exists for the selected group or group member.

If you first select a group or specific managed element, the Relationship Viewer opens with the selected element(s) centered and with the radius slider visible. If you do not make such a selection first, the Relationship Viewer opens with all managed elements visible at the lowest zoom level.

If multiple relationships are available, a sub-menu of your choices is displayed when you click Launch Relationship Viewer; click your preference to open the Relationship Viewer using that relationship.

Adding the Relationship Viewer to a Portal page

You can add the Relationship Viewer to any page in your portal in the same way you add any other application. See Adding a Page (see page 20).

Adding a Web Content application to a Portal page

Using the UMP WebContent portlet, you can create a URL that opens the Relationship Viewer. This lets you place multiple Relationship Viewers on a single portal page, which can be useful if you want to have separate views for different contexts (say, Building-1 and Building-2). See Launching the Relationship Viewer using the WebContent Portlet (see page 265) for details.
Using the Relationship Viewer

This section shows you how to use the controls and functionality provided in the Relationship Viewer. When using the Relationship Viewer, you have two primary kinds of actions:

■ Changing what you see; that is, what is visible and how it appears.
■ Finding objects and seeing details about them.

Changing What You See

There are several ways to change what you see in the Relationship Viewer:

Navigate within the picture

You navigate within the view by zooming in on a part of it, or by panning the view port to various areas when the entire view is too large to fit within the boundaries of the window.

■ To zoom in or out, you can do any of the following actions:
  ■ Drag the Zoom slider right or left to zoom in and out, respectively.
  ■ Drag the handle of the view port indicator in the Radar View.
  ■ Use the mouse wheel (providing you have one).

■ To pan across a view that exceeds the size of the view port, you can either drag the view port indicator in the Radar View or use the scroll bars along the edges of the view port.

There is also a "magnifier" tool that simulates a circular magnifying glass that follows your mouse over the view. This can be helpful when a region of the view is especially detailed.
Change the Layout Style

There are several kinds of layouts available:

- Circular
- Organic
- Orthogonal
- Hierarchic

Each one displays the same managed elements and relationships, but different layouts are more natural for different situations.

Change the Relationship

If the underlying applications model more than one type of relationship, you can select which type of relationship you want to view for the current managed elements.

Change the Radius

If the Relationship Viewer is launched with the context of specified elements (including groups of elements), it includes the "Radius" slider. Such context is most frequently seen when launching the Relationship Viewer from the Unified Service Manager (USM) portlet. The Radius slider lets you adjust the radius of relationships shown from those elements.

All of these are accomplished with controls in the Relationship Viewer control bar.

More information:

- Search and Clear Search Controls (see page 261)
- Radar View Button (see page 258)
- Change Layout Style Selector (see page 258)
- Magnify Button (see page 260)
- Zoom Slider (see page 260)
- Radius Slider (see page 261)
- Relationship Selector (see page 261)

Relationship Viewer Control Bar

The controls for the Relationship Viewer are all located at the top of the screen.

Show/Hide Radar View (button)

Toggles the radar view (an overlay with a small representation of the whole) on or off.

Layout Style (selection list)

Lets you select which style of layout you prefer.
Using the Relationship Viewer

Chapter 18: Relationship Viewer

Magnify (button)

Toggles the magnifier (a simulated magnifying glass that follows the mouse) on or off.

Zoom (slider)

Dragging the indicator right zooms in on the view; dragging it left zooms out.

Radius (slider)

Visible only when the Relationship Viewer is launched with a context (system, or group), dragging this slider changes the radius of visible relationships.

Relationship (selection list)

Lets you select which kind of relationship you want displayed, providing that the underlying applications support multiple types of relationships.

Search (button)

Activates a search for the entity in the text field.

Search entry (field)

Enter an identifier for the managed element you want to locate, and click the "Search" button.

Clear

Clears the search field, removes the "matched element" glow, and resets the zoom to show everything.

Get Link URL (icon: 

Opens a small window that displays the URL of the current view. You can copy this and use it as the basis for a building a URL to use in the WebContent portlet, or to launch the Relationship Viewer from an arbitrary browser. See Launching the Relationship Viewer using the WebContent Portlet (see page 265) for details.

More information:

Radar View Button (see page 258)  
Change Layout Style Selector (see page 258)  
Magnify Button (see page 260)  
Zoom Slider (see page 260)  
Radius Slider (see page 261)  
Relationship Selector (see page 261)  
Search and Clear Search Controls (see page 261)  
Get Link URL Button (see page 264)  
Search and Clear Search Controls (see page 261)
Radar View Button

The Radar View, a small representation of the whole view, is the key to easy navigation within the Relationship Viewer. It is an overlay in the top left corner of the main window, as shown below.

Use the "Show/Hide Radar View" button to alternately show or hide the Radar View.

Note the shaded rectangle within the boundary of the Radar View. It shows the dimensions and location of the current view port with respect to the main view.

You can click anywhere within the shaded rectangle and drag it to a new region of the main view. The view port reflects the change as you drag.

Also note the small black square, or handle, at the lower right corner of the view port locator. You can drag that handle to resize the view port indicator, which at the same time zooms the view in the main window accordingly.

Change Layout Style Selector

This selector looks like a labeled button, with a drop-down arrow on the right-hand end: [Labeled Button]. The label indicates the current layout style.

Click the drop-down arrow to open a list of available layouts. Each layout style displays the same managed elements and relationships, but different layouts are more natural for different situations.
About Layout Styles

This section describes the various available layouts and their uses.

Circular Layout

A circular layout emphasizes group and tree structures among related elements, arranging significant groups as separate circles. These separate circles are presented in as a radial tree, with connections between them.

This layout emphasizes group and tree structures within the relationships.

Hierarchical Layout

A hierarchical layout is primarily used when the relationships being modeled are directed. The elements are placed a hierarchy, and most relationship lines are similarly oriented. For example, the lines may generally run from top-to-bottom.

The hierarchical layout style is good for showing dependency relations between elements, particularly any chain of dependencies between the elements. It is not a particularly good layout for symmetric relationships, such as network connections.

Organic Layout

An organic layout is a general purpose layout for symmetric relationships. It can be used with complex networks, including enterprise networking or system management.

In essence, elements are treated as physical objects with a repulsive force, while connections are treated as springs attached to each pair of elements. The springs produce repulsive or attractive forces between their end points depending on their degree of stretch or compression. In the end, this layout arranges the elements in a way that balances all these pseudo-forces.

Depending on the relationships involved, this layout can help you visualize inherent symmetries and clusters within the data, and show a well-balanced distribution of elements with a minimum of edge crossings.

Orthogonal Layout

An orthogonal layout produces a "bus-oriented" layout. Common connection lines are shown in parallel. This layout can be used to visualize commonalities across relationships.
Magnify Button

The Magnifier simulates a circular magnifying glass on the view port, as shown below. You can click on an element or relationship to view its details panel even when using the magnifier.

Use the "Magnify" button to turn the Magnifier on or off.

Zoom Slider

Drag the Zoom control—illustrated here—to zoom in or out of the main view.
Drag the indicator left zooms out; dragging it right zooms in.

**Radius Slider**

Drag the Radius control — illustrated here — to set the maximum radius visible from selected elements in your view.

![Radius Slider](image)

**Note:** Because radius is based on hops from specific managed elements, this control is only available if you select a specific managed element or group of elements when opening the Relationship Viewer.

**Example:**

Providing you have the CA Nimsoft topology application installed, one useful application of this control is to launch the Relationship Viewer with a context such as, say, "Servers". Setting the radius to zero places all the servers in the view. By incrementally increasing the radius, you can observe how and where their connections are related.

**Relationship Selector**

This selector looks like a labeled button, with a drop-down arrow on the right-hand end: ![Relationship Selector](image). The label indicates the current relationship being depicted. For example, the CA Nimsoft infrastructure models network topology; in this case, the label is **Network Connectivity**.

If NSM applications that create elements and relationships model multiple different relationships, click the drop-down arrow of the Relationship Selector to choose which type of relationship you want to view.

The availability of different relationships depends on what additional applications are present in your installation.

**Search and Clear Search Controls**

You can easily locate any managed element and center it in the view by performing a search on its name. The **Search** button, search entry field, and **Clear** button appear in that order near the right-hand end of the Relationship Viewer Control bar:

![Search Controls](image)
To search for specific managed elements

1. Enter an identifier for the managed element in the text entry field beside the "Search" button. Valid identifiers include the following:
   - Name
   - Class
   - State
   - IP address
   - MAC address.

You can use a wildcard character (*) at the beginning, end, or both ends of the search text. The following are all valid search parameters:

- **MySwitch** — finds the element with the name "MySwitch", if it exists.
- **Switch** — finds any and all elements that end with the string "Switch".
- **building1** — finds any and all elements that contain the string "building1"
- **My** — finds any and all elements that begin with the string "My".
- **10.0.4.6** — finds the element with the address "10.0.4.6", if it exists.
- **10.0.4.*** — finds any and all elements whose addresses begin with "10.0.4".
- **.6** — finds any and all elements whose addresses end with "6".
- **.4.*** — finds any and all elements that include a field in their address with the value "4"

**Note:** You can only use a wildcard character at the beginning or end of a search term (or both ends). You cannot embed one within a search term. For example, the following is an invalid search term: **10.*.4.***
2. Click the **Search** button. If a managed element by that name exists, the Relationship Viewer zooms in to it. If more than one element matches the search, the Relationship Viewer zooms in to a point where all elements are visible in the view port. All elements that match the search are highlighted with a green glow. For example, entering `*.4.6` in the search entry field and then pressing the **Search** button would yield the following:

![Diagram](image)

If no managed element matches the search criteria, the search entry box turns red: 10.0.4.125. No objects are highlighted as a result of the search.

3. Subsequent searches after the first add new elements to the set of highlighted elements; that is, existing highlights remain. To remove the highlights, press the **Clear** button to the right of the search entry field.

**Note:** The search function examines all relevant identifiers in the database to find a match. This means it may find a match on a subordinate identifier, not the one used to label the object in the view. This is, of course, a valid match but may not appear so at first.
Get Link URL Button

At the extreme right edge of the Relationship Viewer, you will notice the Get Link URL button:

Click this button to open a small window that contains a template URL to the current view. You can use the Copy to Clipboard button ( ), if applicable in your environment, to capture the URL. You can use the template URL as the basis for building a functional URL to use in the WebContent portlet, or to launch the Relationship Viewer from an arbitrary browser.

The template URL does not include authentication information. See Launching the Relationship Viewer using the WebContent Portlet (see page 265) for information on how to add authentication information to the template, so that it becomes a usable URL.

Viewing Object Details

By clicking on a managed element or relationship line in the Relationship Viewer, you can get further details about the element or relationship. The specific details depend on the underlying application, and will vary accordingly.

Managed Element details

Click on any managed element to display a panel of information about that element.

This panel includes the following information:

- The primary name of the element.
- A symbol for the element, chosen to represent its type.
- A block of information about the element, including its class, manufacturer, and type.
- A logo that quickly identifies the manufacturer of the element.
- A button that lets you quickly access any alarm information about the element.
- A state indicator, which shows you the current state of the element at a glance.

At the bottom of the panel there is an arrow: . Click this arrow to display a table of the elements properties and values according to data stored for it.

Relationship details

Click on any relationship line that connects managed elements to display a panel of information about that relationship.
This panel includes the following information:

- The type of the relationship.
- A symbol representing the status and type of relationship.
  - This symbol indicates a symmetric relationship:
  - This symbol indicates a directional relationship:
- For each of the related elements, the panel includes the following:
  - A symbol chosen to represent the element's type.
  - The primary name of the element.
  - The class of the element.
  - Port information for each element with respect to the relationship. In this instance, the word "port" should be taken to be generic. That is "port information" is information about the means by which the element is related to the other.

More information:

Changing What You See (see page 255)
Relationship Viewer Control Bar (see page 256)

Launching the Relationship Viewer using the WebContent Portlet

This section shows you how to form a URL that will launch the Relationship Viewer using the WebContent portlet.

Note: A URL constructed according to the information in this section can be used to open the Relationship Viewer in any browser at any location. For example, you could use such a URL from your computer at home. To accomplish this, simply prefix the Source URL as defined below with http://<host>[:port]. You can use the link icon in the Relationship Viewer to obtain most of the URL; remember that the username and userspass parameters are mandatory. You will have to add them after copying the URL. See the Relationship Viewer Control Bar (see page 256) topic for further information.

First, add a Web Content application to your target page. Once the Web Content application is added, a link prompts you to configure it. Click the link to open the the WebContent editor, where you will enter the URL information.
The format of a full Source URL is as follows:

/relationshipviewer/jsp/standalone.jsp?type=type&relationship=relationship_name&elements=id&radius=integer&user=username&password=userpass

**Note:** All URL parameters except `username` and `userpass` are optional. The parameters can be in an arbitrary order. Also note that you should use URL encodings for certain characters of the values. See, for example, the [W3schools web-site](https://www.w3schools.com) (not affiliated with CA Nimsoft).

As shown in the above example, the parameters for the URL are separated by an ampersand character —&. They are defined as follows:

- **type**
  - The type of managed element or elements to display.
  - Options: `computer`, `group`, or `configuration_item`
  - Default: `computer`

  **Example:**
  ```
  ...&type=group&...
  ```

- **relationship**
  - The relationship type, e.g., `physical_connection`, which is the relation type used for modeling topology.
  - Other options depend on what other relationships are available with your installed applications
  - Default: `physical_connection`

  **Example:**
  ```
  ...&relationship_name=physical%5Fconnection&...
  ```
  Note that the underscore character is encoded in the value.
Obtaining group IDs and/or device IDs to use in the `elements` parameter(s) requires that you inspect your database to find the necessary identifiers. You will need to look for identifiers of these types:

- `csId`: the unique id of a computer system. `csIds` constitute a column in the `CM_COMPUTER_SYSTEM` table.
- `ciId`: the unique id of a configuration item. `ciIds` constitute a column in the `CM_CONFIGURATION_ITEM` table.
- `grp_id`: the unique ID of a group. `grp_ids` constitute a column in the `CM_GROUP` table.

There can be any number of `elements` parameters, separated by ampersands. That is, you can have multiple `elements` parameters, but each can specify only one `id`.

**Example:**

```
...&elements=256&elements=257&elements=4572&...
```

**user=username**

- `username`: The user’s login identification in clear text. This is a required parameter, with no default value.

**Example:**

```
...&user=TimA...
```

**password=userpass**

- `userpass`: The user's login password in clear text. This is a required parameter, with no default value.

**Example:**

```
...&password=piAlaMode4me
```
radius=integer

integer

The desired radius (number of hops from specified systems or group members) to show in the view.

Options: any integer value.

Default: unconstrained radius.

Example:

...&radius=1&...
Chapter 19: Reports

This section contains the following topics:

- Reports Overview (see page 269)
- Quality of Service Reports (see page 269)
- Service Level Agreement Reports (see page 272)

Reports Overview

There are two kinds of web reports that you can view in the Reports portlet:

- Quality of Service reports, which must be created manually in the Performance Reports Designer (PRD) portlet.
- Service Level Agreement (SLA) reports, which are created automatically after you create SLAs in the Service Level Manager (SLM) portlet.

The Reports application uses a folder structure that groups web reports into the following three sets:

- Custom - The Custom folder contains custom QoS reports from the PRD portlet. Access to the Custom reports requires the Custom Reports ACL permission.
- SLA - The SLA folder contains SLA reports from the SLM portlet. Access to the SLA reports requires the SLM View ACL permission.

You can browse the Custom, Dynamic, or SLA folders and double-click any report to display that report in a separate window.

Quality of Service Reports

This section describes the QoS reports that open when you double-click a report in the Custom or Dynamic folder.

The following items describe how the graph and its features respond to mouse movements:

- When you move the cursor within a graph, two bisecting guidelines appear and follow the cursor. The date and time that the cursor points to in the graph is shown at the top or bottom of the vertical guideline.
If you click and hold, you can drag the guidelines and select an area in the graph. The graph updates and displays the area you selected over the entire area of the graph.

When you move the cursor within the graph, buttons appear, some of which remain grayed out until you hover the cursor over them.

The graphs contain the following buttons:

<table>
<thead>
<tr>
<th>Button</th>
<th>Description</th>
</tr>
</thead>
</table>
| Style icon | Click to choose the style of the chart or to remove a data series from the chart:  
- Line chart  
- Area chart  
- Column chart  
- Removes the selected data series from the chart |
| Choose Source combo box | Click to chart the same set of data series for another host or target.  
If you have selected multiple targets for a single host (QoS Selection by Host), clicking this displays a list of hosts with the same QoS measurements and targets. This allows you to easily view the same set of measurements for different hosts.  
If you have selected a single target for multiple hosts (QoS Selection by Target), clicking this displays a list of targets for the selected QoS measurement. This allows you to easily view different measurements for the same hosts.  
The menu bar displays the name of the current host or target. If there are no other corresponding hosts or targets, the combo box is not displayed. |
| Maximize icon | Maximizes the display of the chart within the portlet window. |
| View CSV data | Click to export the data displayed in the chart as a CSV file. The Chart CSV Preview dialog displays the first 10 records so you can preview the data. From the dialog you can save the data to a file or copy it to the clipboard. |
Generate chart URL

Click to generate a URL that can be used to display the chart from a browser without being logged into UMP. This displays the Chart URL dialog. Click the Copy to clipboard icon to save the URL to the clipboard.

Data range buttons

By default, the y-axis displays the range of data values for the measurement(s) plus or minus 5 percent. For example, if the data values range from 40 to 80, the y-axis displays a range of 38 to 84. However, for some data units, such as percentages, you may prefer to see the data displayed at a different scale. Clicking the button sets the baseline of the y-axis to zero, and clicking the button sets the top of the y-axis to 100 percent. The buttons appear next to the left or right y-axes if appropriate for the data units. If the data unit is percent, for example, the top and bottom buttons are displayed. If the data unit is megabytes, however, only the bottom data range button is present.

Reports opened from the Dynamic folder have a table below each graph, which contains the following fields and information:

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Source</td>
<td>The computer hosting the probe sending the QoS data.</td>
</tr>
<tr>
<td>Target</td>
<td>The device/target being monitored, supplying the probe with QoS data.</td>
</tr>
<tr>
<td>Min</td>
<td>The minimum value for the QoS, if applicable.</td>
</tr>
<tr>
<td>Max</td>
<td>The maximum value for the QoS, if applicable.</td>
</tr>
<tr>
<td>Standard Deviation</td>
<td>Standard deviation of the samples within the report period (using the STDEV() function in SQL Server).</td>
</tr>
</tbody>
</table>
Service Level Agreement Reports

This section describes the Service Level Agreement (SLA) reports that open when you double-click a report in the SLA folder. SLA reports display information about compliance with SLAs and Service Level Objectives (SLOs). SLA reports are created automatically after you create SLAs in the Service Level Manager (SLM) portlet.

The initial page provides the Service Level Objectives (Summary), which lists the SLOs defined in the SLA, and status information, weight information, and an achieved/expected compliance percentage.

Select an SLO from the Objective column to view the QoS constraints within the SLO. The Quality of Service (Summary) lists the QoS constraints defined in the SLO, and status information, source/target, weight information, and an achieved/expected compliance percentage.

Select a QoS constraint from the Object column to see a graph of the current compliance period for the selected QoS constraint.

The Unavailability Report

Below the graph is a report summarizing the periods with threshold breaches or when the service was unavailable.

The table contains the following information:

- State (why the data was unavailable)
- Period start
- Period end
- Minutes (how long the data was unavailable)

The following icons indicate the status:

- 🔴 = breach
- ⚠️ = unavailable service
Chapter 20: Report Scheduler

The Report Scheduler portlet allows you to schedule reports to run at specified times. Reports can be either Performance Reports or SLA Reports. The reports can be delivered as a PDF via email or FTP, or can be stored on a server.

This section contains the following topics:
- Working with Report Scheduler (see page 273)
- Administering Report Scheduler (see page 277)
- Troubleshooting Report Scheduler (see page 282)

Working with Report Scheduler

This section tells you how to work with jobs in Report Scheduler. It tells you how to create, edit, delete, preview, copy, and run jobs. It also tells you how to view the history for a job and how to view the log file for each job run. The jobs table of the Report Scheduler is also described.

Create or Edit a Job

To schedule a report, you create a job.

You can edit any of the settings of an existing job or disable a job so that it does not run.

To create or edit a job:

1. To create a report, click the Create New Job icon ((Create New Job Icon Image)).
   To edit a report, select the report and click the Edit Selected Job icon (Edit Selected Job Icon Image).

2. In the Job Info section:
   a. Provide a name for the report and check the Enabled option to allow the job to run.
   b. If desired, assign the job to an account.
   c. If desired, enter a description for the job.
   d. Select Performance or SLA for the type of report.
   e. Browse to the report you wish to schedule.
Working with Report Scheduler

f. If you selected **Performance** for the **Type**, make a selection from the **Time Frame** drop-down menu.

   If you selected **SLA** for the **Type**, make a selection from the **Depth** (see page 298) drop-down menu.

   **Note**: The **Full SLA** option provides the most detail, but is more resource intensive than the other options.

g. Select the desired **PDF Page** options for the report.

3. In the Schedule section:
   a. Select an option under Scheduling.
      
      The Run this job pane updates dynamically based on your selection.
   
   b. Specify the time and date to run the report. Use the **With Time Zone** drop-down menu to select the time zone for the report. The time zone you select determines when the job is executed, and which time zone the report displays. If you do not select a time zone, the local time zone is used by default.

      **Note**: The job will run at the first specified time after the **Starting** time you set.

4. In the Delivery section, choose at least one delivery method.

   If you select **Email using this template**, note that **SMTP server settings must be configured** (see page 277).

   If you select **FTP with these settings**, enter information in the required fields. You can use the **Location** field to specify a subdirectory in which to save the report. For example, enter **reports/monthly** to save the report in a subdirectory named **monthly**. The **Passive** check box allows you to set passive or active FTP mode. Passive mode is recommended if the UMP server and FTP server are separated by a firewall.

**Preview a Job**

You can view a PDF of a report without sending it to the selected delivery methods by previewing it. This also provides an easy way to save the report on your computer.

**To preview a report**

1. Click the job you want to preview in the Report Scheduler table to select it.

2. Click the Preview Report of Selected Job icon ( ![Preview Report](image.png)).

   A PDF of the report is displayed in your browser.
Run a Job

You can run a job at any time. The report is sent to the specified delivery method(s). Running a job does not affect the regular schedule for the job.

To run a job

1. Click the job you want to run in the Report Scheduler table to select it.
2. Click the Run Selected Job Now icon ( ).
   The job runs and the report is sent to the specified delivery method(s). The report is named <job_name>_<yearmonthdate>_<hoursminutesseconds>.pdf. Any invalid symbols or characters in the job name are removed.

Copy a Job

You can make a copy of an existing job. This is useful if you want to create another job with similar settings.

To copy a job

1. Click the job you want to copy in the Report Scheduler table to select it.
2. Click the Copy Selected Job icon ( ).
   The Set Name of Copy dialog is displayed.
3. Enter a name for the copy of the job.
   If you do not enter a name, the new job is named Copy of <job name>.
4. Click OK.
   The copy of the job is created. You can edit the settings of the copy.

Delete a Job

You can delete jobs that you no longer want. When you delete a job all runs of the job and the job history are also deleted.

To delete a job

1. Click the job you want to delete in the Report Scheduler table to select it.
   You can press Ctrl + click to select multiple jobs, or Shift + click to select contiguous rows of jobs.
2. Click the Delete Selected icon ( ), then click Yes.
   The job is deleted.
**View Job History**

You can view the run history of a job. The history displays the time(s) the job ran and the status of each run. You can also access the log for each job run.

**To view job history**

1. Click the job you want to view the history of in the Report Scheduler table to select it.
   The <Job Name> History dialog is displayed.
2. To view the log for a job run, click the run to select it and click View Log.

**View the Job Log File**

You can view the log file for each run of a job. The log file includes information such as actions taken, times, status, and errors.

**To view the job log file**

1. Click the job in the Report Scheduler table to select it.
   The <Job Name> History dialog is displayed.
2. Click the run you want to view the log for and click View Log.
   The log is displayed.

You can also access the log file from an error message if a job is unsuccessful. The Job Run dialog displays the error message and a link for viewing the log file.

**Report Scheduler Jobs Table**

The Report Scheduler jobs table lists the jobs you have created. To take action on a job, first click on the job in the table to select it.

Click a column heading to sort the list of jobs by that column, and click again to toggle between ascending and descending order. Enter text in the field next to a column header to filter for that text.

The far left column contains icons indicating the status of the job definition:

- ✔ = Enabled.
- ✗ = Disabled.
- ⚠️ = Invalid schedule. This may mean the date specified for running the job does not exist (for example, Feb. 30) or that the date is in the past.
Administering Report Scheduler

This section tells you how to perform administrative tasks for Report Scheduler.

Configure Email Server Settings

To deliver reports via email, Report Scheduler must be configured to use your SMTP server.

Follow these steps:
2. Click the envelope icon (✉️) in the upper right. The Configure Email Server window opens.
3. Enter an email address that your SMTP server can use in the **Email From Address** field.
4. Enter the name and port number of your SMTP server.
5. If desired, select the **Use a Secure Network Connection** (SMTPS) option.
6. Enter a valid username and password for your SMTP server.
7. Click **OK** to commit your changes.

Set Where Reports are Stored on a Server

One of the delivery options in Report Scheduler is to store reports on a server. By default the reports are stored under the root directory `webapps/reportscheduler/archive`, but you can define the root directory for storing reports. If the report is associated with an account, the report is stored in a directory with that account name under the specified root directory.

To set where reports are stored on a server
1. Open Infrastructure Manager.
2. Click on the robot for the primary hub in the tree view.
3. Click the wasp probe to select it in the top right pane.
4. Press CTRL and right-click, then choose Raw Configure from the popup menu.
5. Expand the `webapps` folder in the tree view of the Raw Configure window.
6. Click `reportscheduler` in the tree view. The Report Scheduler keys are displayed in the right pane.
7. Click New Key.
   The New Key dialog is displayed.
8. Enter file_store_root in the Enter key name field.
9. Enter the directory path you want to use as the root directory for storing reports in the Enter value field.
10. Click OK, then click OK again to close the Raw Configure window.
11. Restart the wasp probe.
Default Fonts for Reports

This section provides information about the default fonts you can use for generating reports with Report Scheduler. The default fonts, which are listed below, vary by language and platform. In addition, note that Report Scheduler generates reports using the language in which the report was originally created, which may or may not be the language of subsequent users.

In general, the default fonts below are installed with the corresponding operating system software. However, if you are having trouble with hash signs appearing in your reports, you may not have a default font installed for your language and platform. Use the information below and the troubleshooting topic Unexpected Characters Appear in Reports (see page 284) to determine if you are missing a default font.

If you are missing a default font, simply install one from the list below in the default fonts directory for the operating system. When you restart wasp, the default font is automatically found. You can also use a custom font, although doing so requires further manual steps after installing the font. See the section Set a Custom Font for Reports (see page 281).

For Windows, the default fonts directory is C:\Windows\Fonts. Fonts can also be installed under the Nimsoft installation directory \jre\jre1.6.0_24\lib\fonts.

For reports generated in English, Spanish, or Portuguese, Report Scheduler by default uses a Sans Serif font.

For reports generated in Japanese, Report Scheduler uses one of the following default fonts, provided the font is installed in the default fonts directory for the operating system:

- Windows
  - MS Gothic
  - MS Mincho
- Linux
  - IPA Mincho
  - IPA Gothic
  - VL Gothic
- Ubuntu
  - Sazanami Gothic
  - Sazanami Mincho
  - Kochi Mincho
  - Kochi Gothic
  - Solaris TrueType
- ricoh hg gothic b
- ricoh hg mincho l
- ricoh hg gothicb sun
- ricoh hg minchol sun
- ricoh heiseimin
- ricoh mincho
- ricoh gothic

**Note:** If none of the default fonts above is installed, Report Scheduler will use a Sans Serif font instead.

For reports generated in Chinese, Report Scheduler uses one of the following fonts, provided the font is installed in the default fonts directory for the operating system:

- **Windows**
  - SimSun
  - SimHei
  - MingLiU
  - FangSong
  - KaiTi
  - MS Gothic - supports *some* Chinese characters.

- **Linux**
  - WenQuanYi Micro Hei
  - WenQuanYi Zen Hei

- **Solaris TrueType**
  - Hei
  - Kai
  - Ming

**Note:** If none of the default fonts above is installed, Report Scheduler will use a Sans Serif font instead.
Set a Custom Font for Reports

Most users will want to generate reports using an available default font for their operating system. However, you can manually set a custom font.

Follow these steps:
1. Start Infrastructure Manager.
2. Expand the appropriate hub and select Service to verify that the wasp probe is running.
   - **Note:** If the wasp probe is not running, right-click wasp and select Activate. When wasp starts, continue to the next step.
3. Right-click wasp and select Configure.
   - The wasp dialog opens.
4. Under the Webapps tab, right-click reportscheduler, and select Stop Webapp.
5. Open Windows Explorer and browse to \probes\service\wasp\webapps\reportscheduler\WEB-INF\lib in the Nimsoft installation directory.
   - **Note:** On Windows, the default fonts directory is C:\Windows\Fonts. Fonts can also be installed under the Nimsoft installation directory \jre\jre1.6.0_24\lib\fonts.
6. Copy the file umpreportexport.jar to a temporary folder.
7. Change the file extension of umpreportexport.jar to .zip, and extract its contents to the temporary folder you created.
8. In the files you extracted, open the transforms folder and locate the file prd2fo.xsl.
9. Open prd2fo.xsl in a text editor and do the following:
   - a. Locate the line near the top of the file containing <xsl:param name="font-family-param"> </xsl:param>.
   - b. Edit the line to specify your custom font between the characters > <, as in <xsl:param name="font-family-param">custom_font</xsl:param>.
   - c. Save and close the prd2fo.xsl file.
10. In the files you extracted, open the transforms folder and locate the file sla2fo.xsl.
11. Open sla2fo.xsl in a text editor and do the following:
   - a. Locate the line near the top of the file containing <xsl:param name="font-family-param"> </xsl:param>.
   - b. Edit the line as follows to specify your custom font: <xsl:param name="font-family-param">custom_font</xsl:param>.
   - c. Save and close the sla2fo.xsl file.
12. In the temporary folder you created:
a. Create a .zip file with the folders com, META-INF, and transforms, ensuring that these folders are at the top level of the .zip file.

b. Rename the .zip as umpreportexport.jar.

13. Overwrite the old umpreportexport.jar by copying the one you created to probes\service\wasp\webapps\reportscheduler\WEB-INF\lib in the Nimsoft installation directory.

14. Restart the reportscheduler webapp.

15. Restart the wasp probe.

Troubleshooting Report Scheduler

This section tells you how to troubleshoot common issues with Report Scheduler.

Report Scheduler: Removing an Account User’s Access

In the Reports Scheduler portlet, an account user can still run an SLA report that was moved from their account to an account they do not have access to. In this case, the account user cannot edit the SLA report, but can view and run the SLA report.

To completely remove an account user’s access to an SLA report you must:

■ Move the SLA report to an account the account user cannot access
■ Move the job to an account the account user cannot access

After an administrator moves the SLA report and the job, the account user can still see and run the SLA report until the page is refreshed.
Report Scheduler Shifts Time Axis in PDF Format

In the Performance Reports portlet, you can create a report and then view it as a PDF. You can create the same report with Report Scheduler and view it as a PDF, but the two versions of the same report may appear different. The actual data displayed in each report is the same, but the x-axis of the Report Scheduler report may appear slightly shifted. This occurs because different technology is used to produce PRD and Report Scheduler reports.

In addition, depending on the time scale of the report, the x-axis shift in the Report Scheduler report may be more or less noticeable. For example, if the time scale is in minutes, any x-axis shift in the Report Scheduler report may not even be visible. However, if the time scale is in days, the x-axis shift in the Report Scheduler report may be more noticeable.

Report Scheduler Does Not Display Estimated Values

When you generate a report with Performance Reports, the chart uses bars to display estimated values for periods without data. However, if you use Report Scheduler to email or print the same chart, the chart does not appear the same. This is because charts created with Report Scheduler do not use bars to display estimated values for periods without data.

Java Exception Displayed When Scheduling Reports

Symptom:
When I try to run a job it fails, and when I view the log file a java exception error is displayed.

Solution:
There may be a problem delivering the report. This could be due to an invalid delivery setting, such as FTP server information, email address, or server path. Check the log file for more information. If you see "UNABLE TO EMAIL REPORT!" before the java exception message, you may need to configure UMP for email or the email address may not be valid.

If you see "FTP Reply: 550 550 can't access file." in the java exception message, one of the following problems may be occurring:

- The disk on the FTP server is full.
- The specified FTP user does not have permission to write to the directory.
- The file name contains characters that cannot be saved on the FTP server. Make sure the job name contains only alphanumeric characters.
Some Scheduled Jobs are Not Running

**Symptom:**

Some scheduled jobs are not running.

**Solution:**

If you have multiple large jobs running at the same time some jobs may time out. By default, jobs try to execute for 5 minutes. If the job fails to execute within 5 minutes, that job is skipped until the next scheduled run. If this occurs, you can increase the timeout interval. To do so, change the value of the org.quartz.jobStore.misfireThreshold setting in the WEB-INF/classes/quartz.properties file.

You can also try increasing the number of threads that are allowed to execute jobs.

**Note:** Increasing the number of threads used to execute jobs increases the resources used by UMP and could cause performance problems.

To increase the number of threads used to execute jobs, edit the org.quartz.threadPool.threadCount setting in the WEB-INF/classes/quartz.properties file. The default value is 6.

Unexpected Character Appears in Reports

**Symptom:**

When I generate a report with ReportScheduler, the pound, or hash sign (#) appears in place of normal characters.

**Solution:**

This occurs if your operating system does not have a default font installed for your locale. If this is the case, glyphs that are missing are replaced with hash signs, and the log file provides a message similar to **Glyph "?" (0x4e2d) not available in font "Times-Roman."**

Verify that you are using one of the default fonts for your locale described in Default Fonts for Reports (see page 279). If necessary, install a default font and then restart wasp so that the new font is located.
Note Icon Does Not Display in Report Scheduler Reports

In the SLM portlet, you can define SLAs and SLOs, and optionally attach notes to them. When you view an SLA or SLO with an attached note in the SLA Reports portlet, a note icon appears on the same line, indicating an attached note.

However, if you use Report Scheduler to run a report that includes an SLA or SLO with an attached note, the note icon does not appear in the report.
Chapter 21: Service Desk

CA Nimsoft Service Desk is a full-featured service management solution. Service Desk provides action-based workflows in accordance with ITIL standards that allow you to coordinate incident response and proactive IT management.

Service Desk integrates data from CA Nimsoft Monitor to allow you to unify service management processes with operational monitoring. The Unified Console, for example, displays the My Tickets view of Service Desk tickets and the NM Alarm Console on a single page.

NM and Service Desk data are also integrated in the areas of:

- Incident creation from events (alarms)
- Alarm closure when tickets are closed
- CMDB data
- Reporting

To access detailed information about how to use Service Desk, click the Product Documentation link under Home in the navigation pane.

**Note:** Unlike other UMP portlets, Service Desk must be displayed on a page by itself. It cannot be displayed on a page with other portlets. The My Tickets portlet, however, can be displayed on the same page with other portlets.

This section contains the following topics:

- Create Tickets from Alarms (see page 288)
- CMDB Integration (see page 289)
- Reports with Integrated Data (see page 289)
- Users in Service Desk (see page 290)
Create Tickets from Alarms

Service Desk incidents can be created manually or automatically from alarms generated by NM.

If a Service Desk ticket that is associated with an alarm is closed, the related alarm is also closed.

You can see the relationship between alarms and tickets by using the ID numbers in each. NM alarms contain the SD Ticket number in the Details window (double-click on an alarm in the Alarm Console to open the Details window). Service Desk tickets contain the Alarm ID in the Additional Information tab.

Automatically Create Tickets from Alarms

Typically when Service Desk is configured it is set up to automatically create incidents from NM alarms.

For information on how to do so, see the section Install and Configure the Service Desk Gateway.

Manually Create an Incident from an Alarm

You can manually create a Service Desk incident from an alarm (event) in the Alarm Console. This allows you to easily assign the alarm to the appropriate workflow in Service Desk.

To manually create an incident from an alarm

1. Right-click on the alarm in the Alarm Console and choose Assign from the popup menu.
2. Click nimsoft_service_desk in the Assign To dialog.
3. Click Assign.
CMDB Integration

The CA Nimsoft CMDB Gateway probe and Service Desk Adapter automatically discover and record the attributes and relationships of devices monitored by NM into the Service Desk CMDB. This allows Service Desk tickets to provide additional information about relationships and processes, including incidents, problems, known errors, changes, and releases, and to apply automatic workflow rules.

The information in the Service Desk CMDB is synchronized with NM data; when information about a device is updated in NM or a new monitored device is discovered, the Service Desk CMDB is automatically updated.

For information on configuring the CMDB Gateway probe and the Service Desk Adapter, see the Service Desk Integration Guide.

Reports with Integrated Data

Service Desk comes with preconfigured Unified Reports that combine data from NM and Service Desk. The following reports can be found in the Service Desk folder of Unified Reports:

- **Top 10 Applications by Events and Incidents** - Displays the top 10 applications defined in NM with the most events (alarms) and incidents for the specified time period.
- **Top 10 Servers by Events and Incidents** - Displays the top 10 servers defined in NM with the most events and incidents for the specified time period.
- **Top 10 Network Devices by Events and Incidents** - Displays the top 10 network devices defined in NM with the most events and incidents for the specified time period.
- **Top 10 Devices by Mean Time to Repair vs. Service Quality** - Displays the top 10 devices defined in NM with the highest mean time to repair (the time between the start and close of the event) and service quality (the time between when incidents related to the event were created and resolved) for the specified time period.
- **Top 10 Accounts by Service Desk Incidents** - Displays the top 10 accounts defined in Service Desk with the most incidents for the specified time period.
- **Top 10 Accounts by NM Events** - Displays the top 10 accounts defined in NM with the most events for the specified time period.
Users in Service Desk

In order to use Service Desk, you must log in with a user name that matches the name of a user in NM. In addition, the NM user must have the Service Desk ACL assigned to it.

The following sections tell you how to create users in NM and Service Desk.

Create NM Users

This section explains how to create NM users. The user names must match the names of the Service Desk users you will create in the next section.

**Note:** The NM users must have the Service Desk ACL assigned to them.

**To create NM users**

1. Click Security, User Administration in Infrastructure Manager.
2. Right-click in the User Administration window and click New User.
3. Enter information in the User, Full Name, and Description fields.
   **Note:** Avoid creating Nimsoft users and LDAP users with identical user names.
4. Choose the appropriate items from the Access Control List and Default Profile pull-down menus.
   Be sure to assign the Service Desk ACL to the user. You can create a new profile for the user if needed. For more information on creating profiles, see the Infrastructure Manager online help.
5. Click Set Password, enter a password for the user, and click OK.
6. Click OK in the New User dialog.
   The user is created.

Create Service Desk Users

This section explains how to create a self-service user for Service Desk. The user name must match the name of an NM user created in the previous section.

**To configure a Service Desk user account**

1. Click the Manage Contacts link in the Application Setup Section of the Service Desk Navigation Panel.
   The Manage Contact form is displayed.
2. Enter information in the First Name and Last Name fields.
   This will be the Display Name for the client. Both fields are required.
   **Note:** The user name must match the name of an NM user created in the previous section.

3. Enter additional information if desired, or leave the rest of the contact record form blank.

4. Click Apply Changes.
   The new contact record is created.
   The Contact Details, Location, Open Items, and other tabs are now displayed on the form. The Enable Login check box, Out of Office check box, and other related fields are enabled.

5. Click the Enable Login check box in the contact record section.

6. Relate the user to an organization in the Location tab of the contact record.
   This will be the primary organization of the user. You can relate multiple organizations, but only one organization can be marked as primary.

7. Do the following in the Application tab of the contact record:
   ■ Assign System User ID for the End User
   ■ Assign License type as Self-Service
   ■ Check the Disable Service Feedback check box as this contact is not needed to participate in the Service Feedback process.

8. Click Apply Changes. The user account is automatically associated with the Self-Service group and can access the Service Desk modules allocated to such users.
Introduction to SLA Reports

The SLA Reports portlet application displays performance information for service level agreements (SLAs) defined in the SLM portlet application.

An SLA is an agreement between a client and a service provider that specifies the service to be provided, times, priorities, responsibilities, guarantees, and warranties. SLAs consist of one or more service level objectives (SLOs), which are specific measurable characteristics of the SLA such as availability, throughput, frequency, response time, or quality.

SLOs are composed of one or more QoS objects that are combined to produce the SLO achievement value. For example, an availability SLO may depend on multiple components, each of which has a quality of service (QoS) availability measurement.

QoS objects specify the source, target, threshold, and operating period settings for the QoS measurement.

SLA Reports displays performance information for all three levels of the SLA. When you first open an SLA report, you see information about the SLA as a whole, including compliance levels for each SLO. Click the name of an SLO to view the QoS objects defined for the SLO and compliance information for each QoS object. Click the name of a QoS object to see a graph of the data collected for that QoS object during the current operating period.

You can also view historical information for the SLA. Information in any of the windows can be viewed as a PDF, which can be saved or printed.

If you do not have the SLM View permission set in the Access Control List (ACL), you will see a "Permission Denied" message when you try to run the SLA Reports portlet application.
Viewing SLA History

The SLA History chart shows historical information about the SLA and provides an easy way to view the same SLA report for different periods.

1. Click the History icon.

The SLA History chart is displayed.

Compliance periods where the SLA objectives were met are shown as green columns, while compliance periods where SLA objectives were not met are shown as red columns. Hover over a column to see a pop-up window with the dates of the compliance period and the compliance percentage. The trend line for the data is shown in blue, and the compliance objective is shown as a red line. Click the Maximize icon to enlarge the SLA History chart to the size of the portlet window.

2. Click a column in the history chart to view the SLA report for that compliance period.
Understanding SLA Reports

SLA reports display information about compliance with service level agreements (SLAs) that are created in the SLM portlet.

The report tells you the compliance period, the goal, and the compliance percentage for the current period. It also lists the service level objectives (SLOs) that are part of the SLA and the percentage of fulfillment for each SLO.

To view information about an SLO, click the name of the SLO.

Viewing SLA Reports

1. Open the SLA Reports portlet. Or, if the portlet is already running, click the Open SLA icon.

   ![SLAs dialog]

   You see the SLAs dialog.

   **Note:** If you do not have the SLM View permission set in the Access Control List (ACL), you will see a "Permission Denied" message when you try to run the SLA Reports portlet application.

2. Click the name of the SLA you want to view a report for.

3. Click OK.

   The SLA report is displayed.
SLOs

An SLO is a service level objective, which is a specific measurable characteristic of the service being monitored. SLOs are composed of one or more QoS objects. One or more SLOs makes up an SLA.

To view information about an SLO, in an SLA report click the name of the SLO. You see the SLO window.

The SLO window provides compliance information about the SLO. It also provides information about each QoS object defined in the SLO and fulfillment information for each QoS object.

To view a graph of data collected for the QoS object, click the name of the QoS object.

QoS Objects

QoS objects are the smallest component of an SLA. QoS objects specify the source, target, threshold, and operating period settings for the QoS measurement. One or more QoS objects make up an SLO, which in turn make up an SLA.

To view a graph of data collected for a QoS object, in an SLO window click the name of the QoS object. You see the QoS object window.

If there was a compliance breach for the SLA, or if the probe was unable to collect data, you see an Unavailability Report below the graph.

The Unavailability Report lists the periods for the breach or lack of data. The State column indicates the cause for the unavailability:

= breach

= unavailable service
Preferences Dialog

Access the Preferences dialog by clicking the Settings icon (하겠습니다), then clicking Preferences.

<table>
<thead>
<tr>
<th>Field or Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>List/Report/SLA Name field</td>
<td>Choose the name of the report you want to open by default in this instance of the portlet. If no report name is selected here, the portlet window contains no data when opened. After choosing a report and saving your preferences, log out and log back into UMP to see the specified report displayed by default.</td>
</tr>
<tr>
<td>Period field (Only for Performance Reports)</td>
<td>Choose the time interval to display for the default report.</td>
</tr>
<tr>
<td>Height field</td>
<td>Enter the height, in pixels, for this instance of the portlet to open to.</td>
</tr>
<tr>
<td>Maximized Height field</td>
<td>Enter the height, in pixels, for this instance of the portlet to display at when the maximize icon (확대하기) is clicked.</td>
</tr>
<tr>
<td>Save button</td>
<td>Click to save your changes.</td>
</tr>
</tbody>
</table>

Generating a Report as a PDF

You can view an SLA Report as a PDF that can be printed or saved.

Follow these steps:

1. Click the View as PDF icon.

The PDF Preferences dialog opens.

2. In the PDF Preferences dialog:
   a. Select the desired orientation and size of the PDF.
   b. Select the depth of the report (see page 298) and click OK.

A new window opens and displays the PDF.
Generating a Report as a PDF

More information:

PDF Preferences Dialog (see page 214)

PDF Report Depth

You can select the depth of the PDF version of an SLA Report. The following reports, in order of increasing depth, are available:

SLA Summary

Provides a one-page report showing when the report was generated, the time zone (if set), the current or historical time period of the report, and compliance information.

SLO Summary

Provides the content of the SLA summary report, the history of the SLA's performance over time, and detailed SLO information. Each SLO is displayed on a separate page.

Full SLA

Provides the content of the SLA summary report, the content of the SLO summary report, and detailed QoS information. Each SLO and QoS is displayed on a separate page. The Full SLA option provides the most detail, but is more resource intensive than the other options.

Note: The time zone displayed in the PDF version of an SLA Report may be different than the time zone displayed in the SLA Reports portlet. The PDF version uses the time zone set in the SLM portlet, whereas the SLA Reports portlet uses the time zone of the user's browser. If a time zone is not specified in SLM during the creation of an SLA, the time zone of the server is used.

Navigating PDF Reports

The PDF version of an SLA Report contains items you can click on to navigate. These items are as follows:

- The SLA summary page contains a table listing the SLOs in the report. You can click on items in the Objective column to jump to a page containing details of an SLO.
- The pages displaying details of an SLO contain a table listing QoS objects. You can click on items in the Type column to jump to a page containing details of a QoS object.
- At the top of each page displaying details of an SLO or QoS object, the heading provides links that allow you to navigate back up to an SLO or to the SLA summary page.
Chapter 23: SLM

The SLM portlet application allows you to quickly define and deploy service level agreements (SLAs). Once SLAs are defined and activated, data is recorded and computed automatically. You can view reports on SLA compliance in the SLA Reports portlet application.

You must have the SLM Admin permission set in the Access Control List (ACL), or you will see a "Permission Denied" message when you try to run the SLM portlet application.

This section contains the following topics:

- **SLM Overview** (see page 300)
- **Using SLM** (see page 311)
- **SLM Database Structure** (see page 374)
- **SLM Interface Reference** (see page 379)
- **SLM Terms and Definitions** (see page 390)
- **Troubleshooting SLM** (see page 394)
Service level management (SLM) is an industry-standard framework used for the primary management of network and application services. SLM uses a hierarchical set of measurable criteria to monitor and ensure the validity of service level agreements (SLAs) between customers and service providers. Among other aspects, SLAs typically define a service provider's hours of operation, maintenance windows, uptime guarantees, timeliness in responding to issues, recovery aspects, and service performance.

SLAs consist of one or more service level objectives (SLOs), which are specific measurable characteristics of the SLA, such as availability, throughput, frequency, response time, or quality. SLOs consist of one or more quality of service (QoS) constraints, which specify the source, target, threshold, and operating period settings, and are combined to produce the SLO achievement value.

To summarize, the components of SLM form the following hierarchy:

- Service level agreements (SLAs)
  - Service level objectives (SLOs)
    - Quality of Service (QoS) constraints

The SLM portlet is designed to help you break down SLAs into SLOs and QoS constraints, allowing you to build powerful, extensible, and measurable agreements with clients.
Data Collection and Compliance Calculation

Data collection is normally performed by QoS-enabled probes dedicated to monitoring and reporting changes and threshold breaches. A QoS-enabled probe, such as cdm (the CPU, Disk and Memory monitoring probe), generates a QoS message each time it checks its objective. This message refers to an object defined in the database, and contains sample data.

The data_engine probe subscribes to the primary hub to receive messages collected by QoS-enabled probes. QoS-enabled probes initiate themselves during startup by sending a QOS_DEFINITION message. data_engine picks up and decodes this message, and then inserts it into the database.

Note: To reduce network traffic, the data_engine probe should be installed as close as possible to the database server, preferably on the same server.

The sla_engine probe handles the data that the data_engine probe inserts into the database. sla_engine performs calculations according to the conditions set for the different SLAs, and writes the results back into the database. Calculation jobs are automatically started and run on a schedule specified in the sla_engine probe UI.

Calculation jobs may also be started manually (see Exporting the QoS Data Series (see page 363)).

Compliance Percentage

A compliance percentage is the percentage of time a QoS constraint is within the thresholds of a defined time period.

The sla_engine probe checks each sample taken in a compliance period. The result is compared and summarized as failed or successful.
In the illustration above, the red line represents the threshold value, the green line represents the average value and the blue line represents the actual sample values.

None of the samples breach the threshold line within the operating periods, which means compliance is 100 percent. The five samples that breach the threshold are outside of the compliance period, which was Monday to Monday, with operating periods every weekday from 08:00 to 17:00.

If, for example, the total number of samples within the operating period were 129, and nine samples were breaching the threshold, 6.98 percent of the samples would be considered out of compliance (9 * 100/129). If this were the only data defined in the SLA, and the SLA required 98.50% or better compliance, the SLA would be breached due to a compliance percentage of 93.02 percent (100 percent - 6.98 percent).

**Calculation Terms and Conditions**

An SLA consists of one or more SLOs, each with one or more QoS constraints.

- Compliance is calculated on each QoS and forwarded to the SLO.
- The SLO handles the received compliance data from the underlying QoS constraints, makes its calculations, and forwards the result to the SLA.
- The SLA handles the received compliance data from the underlying SLOs and calculates the total compliance percentage.

The high-level process is as follows:

1. Each of the QoS constraints compares the data collected from probes with the defined threshold value, and calculates the compliance percentage.
2. The SLO collects the compliance values from the QoS constraints and computes the compliance percentage based on the calculation method selected (selects the best value, the worst value, the average value, etc.).
3. The SLA collects the compliance value from the SLOs and calculates the total compliance value, also based on the calculation method selected.
Calculation Terms and Conditions for the QoS

The QoS reflects the data series measured by the probes. The compliance percentage is calculated for each QoS, and the results are presented to the SLO.

The compliance percentage for a QoS is calculated based on the following settings:

- Threshold value - You define a threshold value for each QoS, and each sample in the data series received from the probe is evaluated to determine whether it meets the threshold.
- Operating period - Defines the time interval when the compliance percentage is to be measured. Data samples from outside the operating period do not influence the compliance percentage.
- Calculation method - Determines the way the compliance percentage is calculated for the QoS.

These settings are set in the Quality of Service Constraints dialog. For detailed information, see Defining QoS Constraints (see page 329).
Calculation Terms and Conditions for the SLO

The SLO receives the compliance calculations from the associated QoS’s. The compliance percentage is calculated on each SLO, and the result is presented to the SLA.

The compliance percentage on the SLO is calculated, based on three different parameters:

- **Excluded period** - You can specify periods, which will not count when the compliance is calculated for the SLO. This may e.g. be periods when the monitored system is planned to be shut down due to maintenance etc. Note that the compliance data received from the QoS’s within this excluded period will not count when the compliance is calculated.

- **Calculation method** - Here you can select between different calculation methods determining the way the compliance percentage is calculated for the SLO.

- **You may select between two different types of calculation methods, Formula or Profile:**
  
  **Formula**
  
  Here you can select a mathematical formula to calculate the compliance percentage, based on the input from the QoS’s:
  
  - Average - Calculates the average value of the input from the QoS’s
  - Best - Looks for the QoS with the best result and selects this result.
  - Sequential - The difference between 100 % and the achieved compliance for each QoS is summarized and then extracted from 100%. Example: The SLO receives the compliance calculations from two QoS’s with compliance of 70% and 90 %. Calculated compliance: 100 % - ((100 % -70 %) + (100 % -90 %)): 60 %.
  
  **Weight** - The possibility to weigh the importance of the different QoS.
  
  **Worst** - Looks for the QoS with the worst result and selects this result.

  **Profile**
  
  A set of configurable custom defined formulas, currently available are:
  
  - **AND**. The values of all samples in all QoS’s are AND-ed, resulting in one compliance value. AND means that all sample values from all QoS’s simultaneously must equal or be better than the threshold value to be compliant. See also the section [Definition of Multi-Series Calculation](#) (see page 305).
  
  - **OR**. The values of all samples in all QoS’s are OR-ed, resulting in one compliance value. OR means that at least one of the sample values must equal or be better than the threshold value to be compliant. See also the section [Definition of Multi-Series Calculation](#) (see page 305).
Calculation Terms and Conditions for the SLA

The SLA receives the compliance calculations from the associated SLOs and calculates the total compliance percentage, based on three different parameters:

- **Operating period** - Defines in which time period the compliance percentage should be measured (e.g. Monday to Friday from 08:00 – 17:00). Data series outside this period does not influence the compliance percentage.
- **Weight** - Possibility to weigh the importance of the different SLOs.
- **Calculation method** - Here you can select a mathematical formula to calculate the compliance percentage, based on the input from the SLOs:
  - **Average** - Calculates the average value of the input from the SLOs.
  - **Best** - Looks for the SLO with the best result and selects this result.
  - **Sequential** - The difference between 100 % and achieved compliance for each SLO is summarized and extracted from 100%.
  
  Example: The SLA receives the compliance calculations from two SLOs with compliance of 70% and 80%.
  
  Calculated compliance: 100 % - ((100 % -70 %) + (100 % -80 %)): 50 %.

- **Weight** - Enables the possibility to weigh the importance of the different SLOs.
- **Worst** - Looks for the QoS with the worst result and selects this result.

Multi-Series Calculation Profile

When defining the calculation settings for an SLO, you have the option of selecting a custom, multi-series calculation profile. See the section [Creating a Calculation Profile](#) (see page 336).

Currently supported profiles are:

- **OR** - Compliant if one of the QoS constraints is equal to or better than the threshold value.
- **AND** - Compliant when all of the QoS constraints are equal to or better than the threshold value.
### Example Using OR

In the above example using OR, at least one of the data series must be equal to or better than the expected value. In the above example, this is achieved, except for the period marked red.

### Example Using AND

In the above example using AND, both data series must be equal to or better than the expected value. This is achieved, except for the period marked red.

### Calculation Examples

The high level process for calculating SLA compliance is:

1. Each of the QoS constraints compares the collected data values from the probes with the defined threshold value and calculates the compliance percentage.
2. The SLO collects the compliance values from the QoS constraints and computes the compliance percentage based on a selected calculation method (selects the best value, the worst value, the average value, etc.).
3. The SLA collects the compliance value from the SLOs and calculates the total compliance value, also based on a selected calculation method.
Example 1: One QoS and One SLO

If using calculation method other than Default for the QoS, see the section Creating a Calculation Profile (see page 336).

Data series for QoS 1: Threshold value 4

4 8 7 1 5 7 4 3 8 7

9 of 10 samples equal to or better than the threshold value gives a compliance value of 90%.

QoS 1
Calculation method: Default.
Computed value = 90%

SLO 1
Calculation method: Doesn't matter.
Computed value = QoS 1 = 90%

Service Level Agreement
Calculation method: Doesn't matter (just one SLO).
Computed value = SLO 1 = 90%.
Example 2: Two QoSs and One SLO

If using calculation method other than Default for the QoS, see the section [Creating a Calculation Profile](#) (see page 336).

Example 3: Two QoSs and One SLO, Using Calculation Method AND or OR

If using calculation method other than Default for the QoS, see the section [Creating a Calculation Profile](#) (see page 336).
Example 4: Two SLOs, Each with Three QoS

If using calculation method other than Default for the QoS, see the section Creating a Calculation Profile (see page 336).

The figure at the bottom of this page shows a SLA with two SLOs, each with three QoS constraints.

Example assumptions:

On the figure below:

- SLO 1: Calculating the compliance percentage from QoS 1, 2 and 3, using calculation method Worst makes a compliance percentage of 70 %.
- SLO 2: Calculating the compliance percentage from QoS 4, 5 and 6, using calculation method Average makes a compliance percentage of 90 %.
The table below the figure shows the total SLA compliance percentage, using different calculation methods for the SLA.

**QoS 1**
- Computed value: 100%
- Calculation method: Default.

**QoS 2**
- Computed value: 100%
- Calculation method: Default.

**QoS 3**
- Computed value: 70%
- Calculation method: Default.

**QoS 4**
- Computed value: 100%
- Calculation method: Default.

**QoS 5**
- Computed value: 90%
- Calculation method: Default.

**QoS 6**
- Computed value: 80%
- Calculation method: Default.

**SLO 1**
- Calculation method: Worst
- Selects the compliance from the QoS Constraints with the worst values.
- Achieved Compliance (worse of QoS1, QoS2 and QoS3): 70%

**SLO 2**
- Calculation method: Average
- Calculates the average compliance from the QoS constraints.
- Achieved Compliance: Average of (100% + 90% + 80%)/3 = 90%

**Service Level Agreement**
- Calculation method: Best
- Selects the compliance from the SLO with the best values.
- Achieved compliance (best of SLO1 and SLO2): 90%
The table shows the SLA compliance percentage for the example above, selecting different calculation methods for the SLA:

<table>
<thead>
<tr>
<th>Calculation method</th>
<th>Achieved compliance</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Average</td>
<td>80 %</td>
<td>The average value of the two SLOs (70% + 90%)/2: 80 %</td>
</tr>
<tr>
<td>Best</td>
<td>90 %</td>
<td>The best value of the two SLOs (70% and 90%): 90 %</td>
</tr>
<tr>
<td>Worst</td>
<td>70 %</td>
<td>The worst value of the two SLOs (70% and 90%): 70 %</td>
</tr>
<tr>
<td>Sequential</td>
<td>60 %</td>
<td>The difference between 100 % and achieved compliance for each SLO is summarized and extracted from 100%: 100 % - ((100 % - 70 %) + (100 % - 90 %)): 60 %</td>
</tr>
<tr>
<td>Weight</td>
<td>82 %</td>
<td>Assuming that the weight distribution between SLO 1 and SLO 2 is set to 40 / 60 for the SLA: (70 % * 40/100) + (90 % * 60/100) = 82 %</td>
</tr>
</tbody>
</table>

**Using SLM**

This section describes how to create and work with Service Level Agreements (SLAs).
Setting Up a Probe to Deliver Quality of Service Data

Let's configure a probe to generate QoS data while monitoring your system. In this example, we use the cdm probe.

Launch the property window of the cdm probe of your choice (assuming that you have Nimsoft administrative rights) from the Infrastructure Manager.

The cdm probe supports QoS for each disk, CPU, Memory, Paging activity and Processor Queue length. All QoS definitions, except for the disk properties, are located under the Advanced tab. Check the ones you are interested in.

The QoS definitions for the disk properties are located under the Status tab. Double-click the disk you are interested in and check the Disk Usage QoS Message (in Mb and/or %) option in the dialog.

Reply Yes when prompted for a restart.

The probe will initially send a QOS_DEFINITION message to the data_engine causing the SLM system to recognize the new QoS Object.

This procedure differs from probe to probe.

Overview of Creating Service Level Agreements

The process of creating SLAs consists of the following tasks:

1. Create a new SLA and define the:
   - Compliance requirement
   - Compliance period
   - Calculation method
   - Excluded periods
   - SLA account to attach to (optional)

2. Create SLOs and define the:
   - Calculation method
   - Excluded periods

3. Define QoS constraints with:
   - QoS object and source
   - Expected QoS value
   - Operating period
   - Calculation method
Creating an SLA

You can create a new SLA in the following ways:

- In the top of the left-hand navigation pane, select New > Service Level Agreement.
- In the top of the left-hand navigation pane, click the icon next to Service Level Agreement and select New Service Level Agreement.

To create a new SLA group, click the icon next to that group in the navigation pane and select New Group.
Service Level Agreement Form

The following section describes the fields in the SLA form:

**Name**

Use this field to provide a short name for the SLA.

**Account**

Use the **Account** drop-down list to assign an SLA to an account.

An account typically defines a customer or organization and its associated privileges. Administrators can create and manage accounts through the **Security** menu in Infrastructure Manager.

**ID**

Each SLA you create is automatically assigned an ID number.

**Description**

Use this field for descriptive text, such as the purpose of the SLA.

**Calculation Methods**

This drop-down menu allows you to determine how compliance input from SLOs is used. You can define periods to exclude from the compliance period, such as maintenance windows, or company holidays. The sample values are calculated based on the option you choose from the following drop-down menu options:

- **Average**
  
  Calculates the average compliance from the SLOs.

- **Best**
  
  Selects the best compliance values for the SLOs.

- **Sequential**
  
  Summarizes the periods when the expected value is not met for all SLOs and calculates the compliance, which is the difference between 100 percent and achieved compliance for each SLO.

- **Weight**
  
  Selecting this option opens the Weight Properties dialog, where you can weight the importance of the different SLOs, either manually or automatically.

  **Note:** If you select **Weight**, the Weight icon to the right of the **Calculation Methods** field is enabled.

- **Worst**
  
  Selects the compliance from the SLO with the worst values.

**Compliance Period**
Click the button to the right of the **Compliance Period** field to open the Compliance Period dialog. This dialog allows you to define the start date and time period for an SLA (including its underlying SLOs and QoS). The system uses your selections to compute the SLA compliance percentage. When a compliance period ends, the system measures its compliance and records it in the database.

SLA compliance periods are often one month, but the following three options are available:

- **Day**
- **Week(s)**
- **Month(s)**

By default, weekly and monthly compliance periods have a fixed start date. For example, if you select **Month(s)**, the **Starting** day is the first day of the month. However, you can change the day the report starts to meet your requirements.

**Timezone setting**

This option within the Compliance Period dialog allows you to set the time zone for your location. By default, the system selects the time zone where the data_engine is located. If the user's location is the same as that of the data_engine, the default setting is normally appropriate. However, you can adjust the timezone setting if you collect data from a location in another time zone.

**Percentage**

This field allows you to set the compliance percentage of the SLA.

**Status**

This message provides the current date and time.

**The current period**

Shows the current period defined in the Compliance Period settings.

**Time used in the compliance period**

A status bar indicating the current point in the compliance period.

**Achieved compliance**

Shows the current compliance percentage values.

**Expected compliance**

Shows the expected compliance percentage values.

**Trend analysis**

Indicates if and when the SLA is breaching the defined compliance percentage. The status bar indicates in red how much of the allowed unavailability has been used. If the status bar is almost completely red, the compliance percentage is close to being breached.
Note: As soon as you enter text or change any selections in the SLA form, the ability to save is enabled in the following buttons:

- The Save icon in the upper right-hand corner
- The **Save** button in the bottom right-hand corner
- When clicked, the small x that closes the form prompts you to save any changes before the form closes.

Objectives (SLO) Tab in SLA Form

The service level objective list shows the status of the objectives within the SLA. The following section describes the fields and buttons under the Objectives (SLO) tab.

**Name**

Use this field to provide a short name for the SLO.

**Description**

Use this field for descriptive text, such as the purpose of the SLO.

**Weight**

This field allows you to weight the importance of the SLO in relation to the other SLOs within the same SLA. The possible values are **auto** or a percentage value.

**Fulfilled**

Shows the percentage of fulfillment of the objective. **100%** means completely fulfilled.

**New [Button]**

Opens a dialog where you can create a new objective within the current SLA.

**Edit [Button]**

Edit the selected objective.

**Delete [Button]**

Removes the selected objective from the current SLA.

**Save [Button]**

Save is enabled as soon as you enter text or change any selections in an SLO.

**Legend**

- The SLO is compliant with the requirements stated by the SLA and with the QoS constraints defined within the SLO.
- The SLO is not compliant.
- No compliance values are currently available.
Alarm Notification Tab in SLA Form

SLM is capable of generating alarms whenever an SLO breaches the defined compliance settings. Like others, these alarms may be forwarded to email, paging, etc.

Alarm when compliance is breached

Check this option to be notified of an SLA breach. A standard alarm is issued when the compliance breaches the value you specify in the **Expected** field.

**Severity Level**

Select the appropriate severity level.

**Alarm Message**

Enter the message to be generated when compliance is breached.

**Subsystem**

Select the subsystem in which alarms will appear in the Alarm Console of Infrastructure Manager.

Alarm when warning threshold is breached

Check this option to be notified prior to an SLO breach.

**Severity Level**

Select the appropriate severity level.

**Alarm Message**

Enter the message to be generated when the warning threshold is breached.

**Save [Button]**

Save is enabled as soon as you enter text or change any selections in the Alarm Notification tab.
Notes Tab in SLA Form

The Notes tab lets you record relevant information pertaining to an SLA. For example, a system administrator can leave a note about an event that affected SLA compliance while he or she was working.

Date

Ties the note to a specific date and time.

Title

Descriptive text explaining the circumstances for the note.

Official

Use this check box to include the note in reports.

Text

The actual body of the note.

New [Button]

Create a note within the current SLA. The Note editor appears for you to type your note.

Edit [Button]

Edit the selected note.

Delete [Button]

Removes the selected note from the current SLA.

Save [Button]

Save is enabled as soon as you add a new note or make changes to an existing note.
Calculations Tab in SLA Form

A compliance calculation is automatically performed on schedules as specified in the sla_engine. However, you can use the Calculations tab to recalculate compliance for specific jobs.

The list shows all available compliance jobs related to the current SLA.

**Job Id**
- The identification number assigned to a new job.

**Period Begin**
- Defines the starting date and time of the compliance calculation period.

**Period End**
- Defines the end date and time of the compliance calculation period.

**Owner**
- The name of the operator that requested the calculation job.

**Job Start**
- Defines the date and time for the compliance calculation job to start.

**Job Expire**
- Defines the date and time to automatically remove the job from the SLM system.

**History**
- Indicates that compliance data should recorded and saved for historic reports.

**New [Button]**
- Opens a dialog where you can initiate a new compliance calculation using the current SLA.

**Edit [Button]**
- Edit the selected job. This is enabled only when the job is queued.

**Delete [Button]**
- Deletes the selected job from the system.

**View [Button]**
- View the result of the compliance calculation for the selected job.

**Save [Button]**
- Save is enabled as soon as you add or make changes to calculations.

Also see the section [Exporting the QoS Data Series](#) (see page 363).
Excluded Periods

The Excluded Periods tab allows you to exclude designated maintenance periods, or other provisioned down time, so that availability data isn't affected. Clicking the New button launches the Exclude Periods dialog where you can specify excluded periods. After you define an excluded period, it will appear in the list that displays when the Excluded Periods tab is selected.

**From date**
Defines the starting date and time for when to exclude data.

**To date**
Defines the ending date and time for when to exclude data.

**Note**
Use this field to add notes about the excluded period.

**Official**
Select this check box to include the Note in reports.

**New [Button]**
Opens the Exclude period dialog, where you can define an excluded period for the SLA selected.

**Edit [Button]**
Allows you to edit the selected excluded period.

**Delete [Button]**
Deletes the selected excluded period.

**Save [Button]**
Save is enabled as soon as you add a new excluded period or make changes to an existing excluded period.

See also the section [Excluding Quality of Service Data](#) (see page 341).
Create a New Service Level Objective (SLO)

You can create a new SLO as follows:

1. Double-click an SLA in the left-hand navigation pane to add the SLO to.
2. With the Objectives (SLO) tab selected, click the **New** button in the Service Level Agreement form.

The Service Level Objective [[example_SLA/New SLO] form appears.

An SLO is built around one or more constrained Quality of Service (QoS) objects. The combination you select of source, target, threshold and operating period settings constrains each QoS object. As with SLAs, you can set alarms for SLO breaches. If necessary, you can exclude an SLO from certain periods, such as scheduled down-time for maintenance.
Service Level Objective Form

The following are fields or messages in the main SLO form:

**Name**
Use this field to provide a short name for the SLO.

**Description**
Use this field for descriptive text, such as the purpose of the SLO.

**Status**
This message provides the current date and time.

**The current period**
Shows the current period defined in the Compliance Period settings.

**Achieved compliance**
Shows the current compliance percentage values.

**Expected compliance**
Shows the expected compliance percentage values.

**Note:** As soon as you enter text or change any selections in the SLO form, the ability to save is enabled in the following buttons:

- The Save icon in the upper right-hand corner
- The Save button in the bottom right-hand corner
- When clicked, the small x that closes the form prompts you to save any changes before the form closes.
The Quality of Service Tab

The following items appear under the Quality of Service tab.

**QoS Name**

The name of the Quality of Service object.

**Source**

Shows the source device of the QoS data.

**Target**

Shows the target device of the QoS data.

**Weight**

Shows the assigned weight of the QoS constraint in relation to the other QoS objects within the selected SLO. The possible values are *Auto* or a percentage value.

**Fulfilled**

Shows the percentage of fulfillment of the QoS constraint. **100%** means completely fulfilled.

**Operator**

The operator code for the value comparison.

**Threshold**

Shows the value set as the expected QoS value.

**Op. Period**

Shows the defined operating period for the QoS constraint.

**Total**

Defines the data samples used for calculations.

**New [Button]**

Create a new QoS constraint within the current SLO.

**Edit [Button]**

Edit the selected QoS constraint.

**Delete [Button]**

Removes the selected QoS constraint from the current SLO.

**Browse [Button]**

Browses the QoS samples related to the current compliance period. You can select to view a graph or table of the QoS samples.

**Save [Button]**

Save is enabled as soon as you enter text or change any selections to QoS definitions.
Move Up, Move Down [Icons 💡, 🔍]

Allow you to change the order of the QoS constraints in the list.

The Quality of Service constraints for the selected SLO show a current state based on the last computations of the SLA Engine. A green indicator shows a positive direction, for example, the QoS is above the SLA compliance percentage. A red indicator means that the QoS is not in compliance with the expected SLA fulfillment. However, this does not necessarily mean that the objective is breached. For the SLO to be breached depends on the weight of the QoS constraint in relation to the other constraints. The legend is as follows:

- The QoS is compliant within the defined constraints (source, target, threshold and period).

- The QoS is not compliant.

- No compliance values are currently available.
Alarm Notification Tab in SLA Form

SLM is capable of generating alarms whenever an SLO breaches the defined compliance settings. Like others, these alarms may be forwarded to email, paging, etc.

Alarm when compliance is breached

Check this option to be notified of an SLA breach. A standard alarm is issued when the compliance breaches the value you specify in the Expected field.

Severity Level

Select the appropriate severity level.

Alarm Message

Enter the message to be generated when compliance is breached.

Subsystem

Select the subsystem in which alarms will appear in the Alarm Console of Infrastructure Manager.

Alarm when warning threshold is breached

Check this option to be notified prior to an SLO breach.

Severity Level

Select the appropriate severity level.

Alarm Message

Enter the message to be generated when the warning threshold is breached.

Save [Button]

Save is enabled as soon as you enter text or change any selections in the Alarm Notification tab.
Notes (SLO Form)

The Notes tab lets you record relevant information pertaining to an SLA. For example, a system administrator can leave a note about an event that affected SLA compliance while he or she was working.

Date
Ties the note to a specific date and time.

Title
Descriptive text explaining the circumstances for the note.

Official
Indicates the note is official.

Text
The actual body of the note.

New [Button]
Create a note within the current SLO. The Note editor appears for you to type your note.

Edit [Button]
Edit the selected note.

Delete [Button]
Removes the selected note from the current SLA.

Save [Button]
Save is enabled as soon as you add a new note or make changes to an existing note.
The Excluded Periods Tab

The service level objective list shows the status of the objectives within the current SLA. The following describes the fields and the action buttons related to SLOs.

From date
Start of the exclusion period.

To date
End of the exclusion period.

Note
The actual text of the note.

Official
Defines the note to be official.

New [Button]
Create a new period within the current SLO.

Edit [Button]
Edit the selected period.

Delete [Button]
Removes the selected period from the current SLO.

Save [Button]
Save is enabled as soon as you make any changes.
The Calculation Settings Tab

The Calculation Settings tab allows you to select how the compliance for an SLO is computed (see the section Compliance Calculation).

The following are fields under the Calculation Settings tab of the SLO form:

Calculation Type

Use the radio buttons to select **Formula** or **Profile**. This selection affects the selections available in the **Calculation Method** drop-down menu.

Calculation Method

The options in the **Calculation Method** drop-down menu depend on the **Calculation Type** you select - **Formula** or **Profile**.

If you select **Formula**, you can select one of the following methods from the drop-down menu:

- **Average**
  - Calculates the average compliance percentage from the QoS constraints.

- **Best**
  - Selects and uses the compliance percentage from the QoS constraint with the best compliance.

- **Sequential**
  - Summarizes the periods when the expected value is not met for all QoS constraints and calculates the compliance. The difference between 100 % and achieved compliance for each QoS is summarized and extracted from 100%.

- **Weight**
  - Weights the compliance from the different QoS constraints according to importance. When **Weight** is selected, the Weight Properties dialog opens, enabling you to set the importance of the different QoS constraints.

- **Worst**
  - Selects and uses the compliance percentage from the QoS constraint with the worst compliance.

If you select **Profile**, you can select one (if any) of the defined profiles in the drop-down menu under **Calculation Method**. Note that these are listed under the **Calculation Profiles** node in the left-hand navigation pane.

**Save [Button]**

Save is enabled as soon as you make any changes.
Defining a Quality of Service Object

Any QoS enabled probe will initiate itself (during startup) by sending a QOS_DEFINITION message. This is picked up by the Data Engine and decoded for information. The content of this message refers to a Quality of Service object in the database. The object will automatically be created in the database with the provided information. You may, however, also create the QOS object manually, by adding the definition using the Service Level Manager. Select File > New> Quality of Service from the menu bar.

The following table describes the various fields in the dialog form:

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>The actual object name on the form QoS_xxx.</td>
</tr>
<tr>
<td>Description</td>
<td>A short descriptive text.</td>
</tr>
<tr>
<td>Group</td>
<td>The group the object belongs to.</td>
</tr>
<tr>
<td>Unit</td>
<td>A string stating the unit (e.g. Milliseconds, Centimeter, etc.)</td>
</tr>
<tr>
<td>Unit abbreviation</td>
<td>Used by reports and views (e.g. ms, cm)</td>
</tr>
<tr>
<td>Has maximum value</td>
<td>Defines the object with a maximum value.</td>
</tr>
<tr>
<td>Is of type Boolean</td>
<td>Sets the type of the object to be Boolean (True/False).</td>
</tr>
</tbody>
</table>

Defining QoS Constraints

The Quality of Service data itself has no value to the service objective unless it is constrained to a specific time-period, specific source/target information and rules to check the actual sample values.

Clicking the **New** button in the SLO form displays the QoS constraint dialog, which has the following fields:

**Object**

The Quality of Service object.

**Source**

Where the sample value originates from.

**Target**

The target name of the sample, such as a disk, URL, etc.

**Expect Quality of Service to be**

Use this section of the UI to set a threshold value for the QoS. Sample values that meet the threshold criteria are considered to be in compliance for the QoS. Select an operator, then a Value and Unit.
Value

The threshold value. Depending on the QoS Object selected, there may be a dropdown menu for this field:

- **Low**
  Smallest value during the operating period.

- **Average**
  Average value for the operating period.

- **High**
  Highest value in the sample range.

- **Maximum**
  The QoS definition states that a maximum value exists for this QoS object, such as a disk sample.

Unit

A QoS definition describes a unit for the QoS value (e.g. kilo, % mS etc). This will be the default unit. For some QoS objects you can select other units. This drop-down list lets you select another unit, if available.

If you change the unit, the value is automatically converted. For example, if the value initially is 1 second and you change the unit to milliseconds, the value automatically changes to 1000.

In operating period

Select the time period for which the compliance percentage is measured. Data from outside this period does not affect the compliance percentage.

- **Always**
  All QoS measurements from within the SLA compliance period (defined in the SLA) are included in the compliance percentage.

- **<Custom>**
  You can create custom operating periods and only data collected during the specified period is included in the compliance percentage for the QoS. If you defined custom operating periods, you see them listed here.

  For example, you could create an operating period named Business Hours for Monday to Friday from 08:00 – 17:00 to measure compliance for the QoS during these hours only.

  For more information, see [Creating an Operating Period](#) (see page 335).
Calculation Method

Select the calculation method to use to determine the compliance percentage for the QoS. You can use one of the formulas listed, or create a custom calculation profile.

Default

Calculates the compliance percentage by finding the percentage of samples that meet the threshold (number of compliant samples/total number of samples).

Example:
Threshold: Greater than or equal to 5
Samples: 5, 4, 8, 6, 2

Three of five samples meet the threshold, so the compliance percentage for the operating period is 3/5 or 60 percent.

Average

Calculates the average of the data sample values and evaluates whether that average meets the threshold (sum of sample values/number of non-null samples).

Example:
Threshold: Greater than or equal to 5
Samples: 5, 4, 8, 6, 2

The total of the sample values is 25, and the number of samples is 5, so the average is 5 (25/5). This meets the threshold, so compliance is 100 percent. If the average does not meet the threshold, the compliance percentage is 0.

Interval

Same as the default method.

Median

Calculated by determining the median value (the value in the middle of the sample range) and evaluating whether that value meets the threshold.

Example:
Threshold: Greater than or equal to 5
Samples: 5, 4, 8, 6, 2

The median value (the value in the middle) is 5. A value of 5 means that the threshold is met, therefore compliance is 100 percent. If the median value does not meet the threshold, the compliance percentage is 0.
<Custom>

You can define custom calculation profiles, which allow you to set advanced properties for the calculation method. If you have defined calculation profiles, you see them listed here.

For example, if you define a custom calculation profile and select **Median** as the calculation method, you can use Variable Configuration to set a floor level and a ceiling level. Values below the floor level and above the ceiling level are ignored when calculating the median value. Or, if you select **Average** as the calculation method, you can set the breach value to a value other than 0 percent, and you can define how null values are handled.

For more information, see [Creating a Calculation Profile](see page 336).

**Percentile**

This option is listed on the dropdown menu if you have created a custom calculation profile using the calculation method **Calculate availability from the N’th percentile of all samples**. For more information, see [Creating a Calculation Profile](see page 336).

The value for a percentile you specify is evaluated to determine whether it meets the QoS constraints. For example, if you specify the 50th percentile, the value that corresponds to the 50th percentile of the data samples is evaluated. If the value meets the constraints, the availability is 100%. If it does not meet the constraints, the availability is 0%.

Use the Variable Configuration table to specify the percentile.

Example:

Threshold: 5

Samples: 5, 4, 8, 6, 2

Percentile: 50th

Availability: First the data is sorted in ascending order: 2, 4, 5, 6, 8. The position in the data array for the percentile is calculated as (number of data samples x percentile)/100. In the example, this is (5 x 50)/100=2.5. However, any decimal places are ignored, so the result is 2. This means the data sample in the second position in the array is used, in this case 4. Since 4 does not exceed the threshold, the availability is 100%.

**Viewing the Current Sample Values**

Use the QoS sample browser (see [Viewing the Quality of Service Data](see page 361)) to determine the best possible values for your QoS threshold settings.
Distributing Importance (Weight)

When setting up service level agreements and objectives, we recognize the fact that some objectives are more important than others. The same fact applies to QoS constraints. The weight distribution feature will help the user to either automatically or manually set up importance (measured in percent) for SLO or QoS constraints.

In Calculations Settings Tab, select Calculation Method as Weight and click the Modify button. Clicking the Modify button opens the weight dialog, showing a pie chart representing the importance (%).

Automatic Weight Distribution

Click button for automatic distribution of weight. The weight will automatically be computed based on the number of objects available (objectives or constraints). This is the default method.

Manual Weight Distribution

This mode enables you to manually distribute weight using the selected object (from the list) and the slider. You can also use the buttons for manual weight distribution. Please note that all of the 100% needs to be distributed. The defined weight will then be displayed in the Weight % column in QoS tab.
Creating a Compliance Period

The compliance period will constrain the SLA (with its underlying SLO and QoS constraints) to a user-defined timeframe, complying with an agreement between two parties. A typical period for SLA monitoring is on a monthly basis. You may, however, create daily and weekly periods as you wish.

The compliance period will, depending on its type (weekly and monthly) have fixed starting dates. If weekly is selected then the starting day is Mondays, and if monthly is selected then the starting day is the first day of the month. However, it is possible to modify the starting day according to your specific needs. E.g. let’s say you want starting day from the 15th; then you adjust the starting date accordingly.

Click the Compliance Period button on the SLA dialog to open the Compliance Period dialog.

See below for a description of the difference in the interpretation of the starting date (at the bottom of the dialog) for the compliance period types.

Day

The starting date is irrelevant.

Week(s)

The starting date defines the starting day in every week.

Month(s)

The starting date defines starting day of every month.

Timezone Setting

This option enables you to set the time zone for your location. By default, the time zone of the location where the data engine is located is selected. As long as the SLM and the data engine are located in the same time zone, this setting should normally be used. If you collect data from another time zone, you can select this time zone here.
Creating an Operating Period

Operating periods are used to constrain the measured values (Quality of Service) to specific hours of any day.

The operating period is defined to be a collection of time periods, and is used when we define new SLAs. We use the Operating Periods when defining the QoS constraints.

Samples falling outside these time specifications will not influence the SLO/SLA compliance requirements.

The Operating Period dialog is opened by clicking the icon beside the Operating Period Node in the Navigation Pane and selecting the New Operating Period option.

Give the operating period a name and an optional description. Click the New button in the dialog to add days and time specifications.

To edit an entry in the list, simply double-click the entry to open the time specification dialog for the selected entry.

For example, let's create an operating period we can call Company Hours. First let's define our work-hours. Normally, they are Monday-Friday, from 0700 to 1700. The scheduled downtime is every Tuesday at 1500-2000 (*).

The "Company Hours" profile will contain a list of time-specifications (days and time) like the one below:

<table>
<thead>
<tr>
<th>Day</th>
<th>From</th>
<th>To</th>
</tr>
</thead>
<tbody>
<tr>
<td>Monday</td>
<td>07:00</td>
<td>17:00</td>
</tr>
<tr>
<td>Tuesday</td>
<td>07:00</td>
<td>15:00 *</td>
</tr>
<tr>
<td>Wednesday</td>
<td>07:00</td>
<td>17:00</td>
</tr>
<tr>
<td>Thursday</td>
<td>07:00</td>
<td>17:00</td>
</tr>
<tr>
<td>Friday</td>
<td>07:00</td>
<td>17:00</td>
</tr>
</tbody>
</table>

* We simply ignore the scheduled downtime because it is outside our "work hours".

The format of the operating period and the time-specifications is so flexible that we can create complex operating periods.
Creating a Calculation Profile

Users are allowed to define their own calculation profiles. This is done by clicking the icon beside any of the Calculation Profiles sub-nodes in the Navigation Pane and selecting New.

- Click the icon beside the SLO Calculation sub-node (or one of the defined SLO calculation profiles), selecting New, opens an empty dialog enabling you to define a new SLO Calculation profile.
- Click the icon beside the QoS Calculation sub-node (or one of the defined QoS calculation profiles), selecting New, opens an empty dialog enabling you to define a new QoS Calculation profile.

These Calculation Profiles can be selected when defining the calculation properties for Service Level Objects (SLOs) and Quality of Service Constraints.

The profiles are based on built-in plug-ins distributed with the Service Level Manager application.

**Name**

Give the new Calculation profile a name in this field.

**Description**

Give a short description of the Calculation profile in this field.

**Calculation**

You can select one of the available calculation plug-ins from the drop-down list. Available plug-ins in the list depends on if you have opened the dialog for a SLO calculation profile (multi-series calculations) or the list for a QoS calculation (single-series calculations).
These plug-ins are available:

For QoS calculations:

- **Calculate availability from the average of all samples.**

  Finds the average value of all samples and compares this value against the defined threshold value.

  If the average value of all samples meets the constraints, the availability for that QoS is 100%.

  If the average value of all samples does not meet the constraints, the availability for that QoS is by default set to 0%. Note: Setting the Breach value to another value, using Variable configuration (see below), and the percentage (on breach) can be set to another value than 0%.

  Also note that you can define the way NULL values are handled.

  (NULL value, typically if a probe does not measure a value from the target due to a timeout, e.g. no answer to a ping).

  Example:

  Threshold: 5

  Samples: 5, 4, 8, 6, 2 and one NULL sample.

  Define NULL sample as ignored and Breach Value as 35%, using the Variables Configuration.

  Availability: The NULL sample is ignored, giving an average value of \((5+4+8+6+2)/5=4.6\).

  This is below the threshold (5), and therefore a breach condition (which we have set to 35%).

- **Calculate availability from number of samples that meet the constraints.**

  This profile calculates the availability by finding the percentage of samples that meet the constraints.

  Note that you may, using the Variables Configuration, define how missing samples should be treated: either ignored (and not influencing the availability) or treated as samples not meeting the constraints.

  Example:

  Threshold: 5

  Samples: 5, 4, 8, 6, 2 and one sample missing.

  Define missing samples as ignored, using the Variables Configuration).

  Availability: One sample missing (ignored), 3 of four samples meets the constraints, giving an availability of 75%.
- **Calculate availability from the median.**

  The median of a number of values is found by sorting the values in a row in descending order. The value found in the middle of the row is the median value.

  Example:

  Threshold: 5
  Samples: 5, 4, 8, 6, 2.
  Sorted in ascending order: 2, 4, 5, 6, 8
  The value in the middle of the row is 5, which means that the median value is 5.

  Availability: The median value 5 means that the threshold value is not breached. This means 100% availability.

  Note: Using the Variables Configuration, you are allowed to define a floor level and a ceiling level. Values below the floor level and above the ceiling level will be ignored and not count when calculating the median value.

- **Calculate availability from the N'th percentile of all samples.**

  The value for a percentile you specify is evaluated to determine whether it meets the QoS constraints. For example, if you specify the 50th percentile, the value that corresponds to the 50th percentile of the data samples is evaluated. If the value meets the constraints, the availability is 100%. If it does not meet the constraints, the availability is 0%.

  Use the Variable Configuration table to specify the percentile.

  Example:

  Threshold: 5
  Samples: 5, 4, 8, 6, 2
  Percentile: 50th

  Availability: First the data is sorted in ascending order: 2, 4, 5, 6, 8. The position in the data array for the percentile is calculated as (number of data samples \times percentile)/100. In the example, this is (5 \times 50)/100=2.5. However, any decimal places are ignored, so the result is 2. This means the data sample in the second position in the array is used, in this case 4. Since 4 does not exceed the threshold, the availability is 100%.
For SLO calculations:

Calculate availability by AND-ing or OR-ing the data series.

Calculate availability by logical expression

For creating a new Multi Expression in SLM, create a new Calculation Profile under SLO Calculations. For setting the logical expression in the profile, set the value of the “Expression” field.

Data Series

Automatic

QoS Data is recorded at intervals as specified in the probe configuration.

Asynchronous

QoS Data is recorded each time the measured value changes.

Variable Configuration

This field enables you to modify the selected plug-in.

Example 1:
If using the plug-in “Calculate availability from the average of all samples”, you double-click the lines in this field to define how to handle Null values samples and Breach values.

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Breach Value</td>
<td>Availability in percentage on breach</td>
<td></td>
</tr>
<tr>
<td>NULL Value</td>
<td>Value used to replace NULL samples</td>
<td></td>
</tr>
</tbody>
</table>

Breach value

When the average value breaches the defined threshold for the QoS, the average value is by default set to 0 % availability for that QoS.
This can be overridden by clicking the Breach value entry and set the Breach value to another percentage, e.g. 54 % as in this example.
Null value

In the case of Null value samples in the data series, you have the following options on how these samples should be treated:

If none of the options described below are selected, a NULL sample will be treated as a not compliant value.

Ignore

The samples will be ignored and will not influence on the compliance percentage. The value “0” will be entered in the Value column.

Min

The samples will be set to the same value as the minimum sample value found in the dataseries. The value “1” will be entered in the Value column.

Max

The samples will be set to the same value as the maximum sample value found in the dataseries. The value “2” will be entered in the Value column.

Note: Click the icon beside the NULL value entry and select Clear Parameter to reset the current parameter value.

Type:

If using the plug-in with AND or OR calculation, you double-click the line in this field to select if the series should be AND-ed or OR-ed.
Expression

If using the plug-in with logical expression, you double-click the line in this field to define the logical expression.

For setting the expression use following guideline otherwise the expression parser error will occur.

- Use AND, OR and NOT operators either in upper and lower case.
- The QOS are represented by integer number.
- Each expression must be enclosed in bracket.
- Each token in the expression must be separated by a space.

Some valid samples of the logical expression are

- \( (1 \text{ or } 2) \)
- \( (1 \text{ AND } 2) \)
- \( (1 \text{ and } \neg 2) \)
- \( (1 \text{ AND } 2) \text{ OR } (3 \text{ AND } \neg 4) \)

Excluding Quality of Service Data

Backup, hardware/software upgrades etc. are normal system administrative tasks that make the systems unavailable for shorter or longer periods of time. Normally these procedures are placed to off-hours, such as evenings and weekends. Scheduled maintenance is covered by the agreement between the service provider and the customer. SLM allows you to exclude data for the period in question prior to the event, or after the event (this requires a recalculation of the SLA compliance).

Excluded periods may be created in two different ways:

- You may create an excluded period for a specific SLA by or SLO.

You may create an excluded period for multiple SLAs by running the Add Exclude Period wizard (see the section Creating an Exclude Period using Add Exclude Period Wizard).
Creating an Exclude Period

You may create an excluded period for a specific SLA or SLO by selecting the Excluded Periods tab in the SLA/SLO dialog and clicking the New button. This opens the Exclude Period dialog. Ticking the Add Note option expands the dialog to display the Note section, which otherwise is hidden.

From

Defines the start date/time for the exclude period.

To

Defines the end date/time for the exclude period.

Add Note

Checking this option expands the dialog to display the Note section, which otherwise is hidden.

The textual note related to this exclude period.

Official note

If this option is checked, the excluded period and the note will appear on the reports.

Title and text

A title and a short text, describing the reason for the excluded period.

You may exclude data for the entire SLA or per SLO. The excluded data will not influence the SLA compliance calculations, and should therefore also be present in the report as a note.
Creating an Exclude Period, using the Add Exclude Period Wizard

You may create an exclude period for multiple SLAs by running the Add Exclude Period wizard. Launch this wizard by selecting Tools > Wizards > Add Exclude Period from the menu bar.

This wizard will add an exclude period with an optional note to one or more SLA/SLO definitions.

The first dialog lets you define when the period starts and stops.

Ticking the Add Note option lets you add an optional title and description of the period. Ticking the Official Note option, the excluded period and the note will appear on the reports.
Click the Next button to continue.

In the next dialog appearing, all SLAs are listed.

If ticking the Show Service Level Objectives (SLO) option, also SLOs will be listed below the SLA they are associated with. Select the components to which you want to add the exclude period.

Click the Next button to continue.

The final dialog appears. Click the Finish button to finish and exit the wizard.
Example - Defining a Simple Service Level Agreement

Important prerequisite:
To be able to measure the service level, you must ensure that the probes monitoring your system are configured to generate QoS (see description in the section Setting Up a Probe to Deliver QoS Data (see page 312)).

The steps involved when defining a simple SLA are as follows:
1. Define your operating periods.
2. Create a new SLA and define your compliance requirement and measure period.
3. Define the Service Level Objectives that make up your agreement.
4. Exclude periods.

Step 1 – Define an Operating Period

The first step is to define an operating period. You will need it in step 3.

The Operating Period dialog is opened by clicking icon beside the Operating Period Node in the Navigation Pane and selecting the New Operating Period option.

Defining no period will default to a 24 hours x 7 days per week service availability, but most organizations have a little more complexity than that. E.g. there may be a regularly scheduled maintenance window when services do not have to be available, or some services may only need to be available during business hours. In the example below, we have chosen to define a “business hours” period.

You can define multiple periods for multiple different services as appropriate.

See also the section Creating an Operating Period (see page 335) for more information.

Defined operating periods will appear in the Operating Periods node in the Navigation Pane, available when defining new SLAs.
Step 2 – Create a new SLA and Define the Service Level Compliance Requirement

Select New > SLA or click icon beside the SLA Node in the Navigation Pane and selecting the New SLA option.

Give the new SLA a name and a description, and set the compliance percentage to 99.9%.

Click the Set period button.

Select Compliance Period

- Day
- Week(s)
- Month(s)

1 week, Starting

July 26, 2010

Choose one of the period intervals as your compliancy period.
This is the period over which the service level will be measured and your committed compliance amount (measured as a percentage) i.e. if you are going to define a service level of say 99.9% availability; over which period will this be measured? You could define this as a daily service level, in which case your SLA compliance will be measured on a day-to-day basis, or it could be measured weekly or monthly. You also define when you want the measurement period to start.

In this case, we have selected a weekly service level that starts on a Monday and ends on Sunday and we are committed to 99.9% compliance.

Leave the Time zone field empty if your SLM is located in the same time zone as the data engine.

Also, see the section Compliance Calculation for selecting a Calculation method different from Average, which is default.

**Step 3 – Define the Individual Service Level Objectives**

Each Service Level Agreement is comprised of one or more Service Level Objectives. Each objective is a data point that we can measure and calculate into the report – the available data points are automatically prompted to the user by using the pull-down tabs.

Make sure the Objectives (SLO) tab is selected and click the New button.

The Service Level Objective window appears. (Note that the section [Creating a Service Level Objective](#) (see page 321) contains additional information).

Give the new Service Level Objective a name and a description. Select the Quality of Service tab and click the New button.

Now the Quality of Service Constraints dialog appears.

In this case we have selected Network Connectivity Response to be less than or equal to 100 seconds and we have selected that this objective should be measured in the "business hours" period that we defined in step 1. We also have the ability to manually “weight” each objective – so that if there are several objectives for us to meet, we can tell the product either to apply equal weightings (automatically selected) or to weight one objective more heavily than others. In this case, we will just select a single objective. See also the section [Distributing Importance (Weight)](#) (see page 333) for more information.

The definition of the Service Level Agreement is now complete.
**Step 4 – Excluding Periods**

Now the SLA setup is complete. But what happens if there is an event outside our control that we want to “exclude” from the SLA calculation? Some examples of this could be that you have agreed with your customers that there will be a one-off maintenance window required for applying some much needed Service Packs. Or, maybe there was a complete power outage to your building that you agree should not count towards the SLA calculation or perhaps a company holiday.

We simply enter the period that we wish to exclude, and add a “note” to that exclusion. Again, by selecting the “official note” box, this note will appear on the reports that everyone gets to see. This feature is extremely useful, because it means that we no longer have to go back and manually recalculate data.

In this case, we have agreed with our web-hosting provider that there was critical maintenance to be performed on our SQL Server database to protect it from the latest Internet worm. We have agreed to take a 2-hour outage that will not count towards the SLA Measurement.

For more information, see the section [Creating Service Level Agreements](#) (see page 312).

**Creating an SLA Using the SLA Wizard**

The SLA Wizard can be launched by selecting the Tools > Wizards Option in the Service Level Manager Toolbar.

Two different SLA wizards are available:
- Creating a SLA based on a service
- Creating a SLA based on an existing SLA

**Example – Creating an SLA Based on a Service**

This wizard will create a SLA group containing one or more SLAs, based on your selections through this wizard (one SLA for each selected source computer). Each SLA will consist of one SLO with one or more QoS constraints, according to your selections.

1. Launch the wizard by selecting Tools > Wizards > Create SLA By Service.
2. First you have to select the type of SLA you want to create. You have two options:
   - Server SLA - Creating SLAs computing data from server related probes.
3. Make your selection and click the Next button to continue.

When selecting source in the wizard, it means:

- The robot hosting the probe for the net_connect probe.
- The device (e.g. a router) for the interface_traffic probe.

In this example we describe a Server SLA.

4. Click the Next button to continue.

Step 1 prompts you for the following information:

**Group Name**

This will be the name of the created SLA group.

**Description**

This is a short informative description of the SLA. This information will be displayed in the Description field of each of the SLAs created through this wizard.

**Account**

Select the Account under which the SLA Group is to be created.

Select Compliance percentage, Compliance period, QoS calculation method and QoS Operating Period as described in the chapter Creating Service Level Agreement.

Optionally attach the SLA to an account.
5. Click the Next button to continue.

Step 2 enables you select the cdm properties for the SLAs. You can select:

- If you want to include CPU usage, Memory usage and Disk usage.
- Which disks to include
- If you want to completely skip the cdm part (selecting Do not include).
6. Click the Next button to continue.

Step 3 enables you select the ntservices properties for the SLAs. You can select:

- One or more of the services listed.
- If you want to completely skip the services part.
7. Click the Next button to continue.

Step 4 enables you to select the processes properties for the SLAs. You can select:

- One or more of the processes listed.
- If you want to completely skip the processes part.
8. Click the Next button to continue.

Finally you can select one or more QoS Source. One SLA will be created for each of the selected QoS sources, and the SLAs will be placed in the SLA group created with the name specified in step 1.

9. Click the Next button to continue.

The wizard is now finished. When you click the Finish button, the SLA group will be created with one or more SLAs, depending on your selections.

Each of the SLAs contains one SLO with one or more QoS objects according to your selections.
Example – Creating a SLA Based on the Selected SLA

This wizard will create a SLA, based on a SLA selected in the Navigation Pane. The underlying SLO(s) and QoS definitions for the selected SLA will be used as a template.

1. Launch the wizard by selecting Tools > Wizards > Create SLA Based On The Selected SLA.

2. Give the new SLA a name and an optional short description.
   - The SLA Description field will initially contain the description of the SLA that we use as basis for the new SLA.
   - The Based on field will contain the name of the SLA you have selected to use as a template.

3. Select an account from the drop-down list at the bottom of the dialog if you want to attach the new SLA to an account, otherwise leave the field empty.

4. Click the Next button to continue.
   - The next dialogs ask you to choose a source and a target for all of the QoS definitions defined for the template SLA.

5. Select your definitions (you may skip one or more QoS objects for the new SLA by checking the Skip this QoS object option).

6. Click the Next button to continue.
   - When all QoS objects are defined, the following dialog appears:

     ![SLM Wizard - Create SLA Based On...] (Image of the final step dialog)

     The final step is to generate the SLA with its underlying SLO and QoS definitions. Press the ‘Finish’ button to finalize this operation.
7. Click the Finish button, and the newly defined SLA will appear in the Navigation Pane.

If the new SLA is based on a SLA in a SLA group, the new SLA will be placed in the same group.

Creating a QoS Monitoring Profile

You can create monitoring profiles that are based on QoS data instead of live information from a system.

The feature will be activated and visible, provided that the qos_engine probe is installed. For each QoS Object you will be allowed to create one or more monitoring profiles of same or different types.

A dedicated probe (the qos_engine) will subscribe to QoS messages, each QoS message will be checked to see if it fits a monitoring profile (a QoS can match to more than one profile). If one or more profiles match the incoming QoS message, the values are evaluated and alarms issued if a threshold violation is detected.

Creating and configuring monitoring profiles can be done by clicking icon beside a QoS object and selecting Create Monitor option under the QoS node in the Navigation pane.

The QoS Monitor dialog pops up, enabling you to set the monitoring properties.

Baseline Value Definition

The baseline value is a central element for the threshold when creating monitors of type Dynamic.

The baseline is calculated from measurements collected at a specific time each day for the last N days, or a specific time for the same weekday the last N weeks.

The samples within all the time spans are used to calculate the average and standard deviation of those values. This gives some fundamental values that can be used for further calculations.

For example, a weekly baseline definition is set to 4 weeks and a time span of 1 hour. The QoS arrives at a Friday 13:00:00. This will, with a sample rate at 5 minutes, give 12 samples for each of the previous 4 Fridays between 12:30 and 13:30. These 48 samples are the basis for the baseline calculation.
QoS Monitors List

Selecting Tools > QoS Monitors in the SLM Menu bar, the QoS Monitors window will be launched. This window lists the QoS Monitors defined (if any).

QoS monitors are created as described in the section Creating a QoS Monitoring Profile (see page 355).

See also the section QoS Monitor Properties (see page 357) for details.

Click icon in the list which gives you the possibility to:

- Edit the properties for the selected QoS monitor (see QoS Monitor Properties (see page 357)).
- Delete QoS monitors.
- View the baseline values for the selected monitor (see Baseline Value Definition (see page 355)).
QoS Monitor Properties

Monitor Name
The monitor name is automatically filled in, based on the QoS name, source and target. This name may be overruled by a name of your own choice.

Monitor Subsystem
This field is automatically filled in with the sub-system ID for the device, identifying from which part which part of the system the QoS and alarms come. The sub-system IDs are maintained by the Nimsoft Alarm Server (nas).

Monitor Types
Select a monitor type to be used as a basis for the threshold calculation. Your choice here depends on what you want to achieve:

Static
Use Static if you want alarms to be issued if the QoS value exceeds the threshold specified.

You may specify the threshold as an absolute value or as a percentage of the maximum value (maximum value is only available for QoS types with maximum size, otherwise grayed out). You may also select that at least n consecutive breaches must occur before an alarm will be issued.

Dynamic
Use Dynamic if you for example want to consider that the QoS values have a regular fluctuation, such as high during the work hours and low at night.

Alarms will be issued if the QoS value exceeds the threshold, calculated from the baseline value.

You must specify Period Type, Periods and Timespan to find the baseline value, which will be used to calculate the error threshold.

- Period Type is day or week.
- Periods are numbers, defining how many days or weeks to be used to calculate the Baseline.
- Timespan is the time window within each period, from which samples should be used to calculate the Baseline value.

The baseline is calculated from measurements collected at a specific time each day for the last N days or a specific time for the same weekday the last N weeks.

The threshold is defined as an expression that is calculated in a Lua script. The expression can contain the following variables that are calculated based on the baseline settings:

- AVG
- STDEV
Using SLM

- MIN
- MAX
- COUNT

Example:
Period Type: Day
Periods: 4
Timespan 1 hour

A sample collected at for example 12.15 o’clock, will be compared with the threshold, which is calculated from the Baseline value. The Baseline value will be calculated based on the average value of the samples collected within the timespan (1 hour) for the four past days.
The threshold can be by either selecting one of the formulas in the drop-down list, such as 20% Above Baseline.

Selecting Custom, you may specify a threshold manually.

Percentile

As for Dynamic (see above), you may use Percentile if you for example want to consider that the QoS values have regular fluctuations.

Alarms will be issued if the QoS value exceeds the threshold, which is a percentile of the values of the samples collected within the time spans for the periods specified.

The threshold can be specified as for example greater than the 95 percentile of the calculated value.

**Alarm thresholds:**

Error thresholds must be defined for the QoS monitors in order to have alarm messages issued on threshold breaches.

For monitors configured to use monitor type Static and Dynamic, it is also possible to define a warning threshold, and you may also select to monitor for unavailable data.

The operands available to be used to set the error- and warning thresholds are:

- Less Than
- Less Than Or Equals To
- Equals to
- Greater Than
- Greater Than Or Equals To

**Threshold values**

- Custom - Selecting Custom, you can compose your own threshold value. If first selecting one of the predefined threshold values, and then selecting Custom
- Baseline - The baseline is calculated from measurements collected at a specific time each day for the last N days or a specific time for the same weekday the last N weeks.
- 10 % Above Baseline
- 20 % Above Baseline
- Baseline + Standard Deviation
- Baseline + 20 % of Stdev
- Baseline Period Minimum
- Baseline Period Maximum
The alarm message text can contain some variables that will be expanded before the alarm is sent. The following variables are supported:

- **NAME**
- **MONID**
- **VALUE**
- **OPERATOR**
- **THRESHOLD**
- **AVG** (dynamic)
- **STDEV** (dynamic)
- **MIN** (dynamic)
- **MAX** (dynamic)
- **COUNT** (dynamic)

A typical alarm message example would be: ${NAME} has breached the threshold value (${THRESHOLD})
Viewing the Quality of Service (QoS) Data

The navigation window contains the Quality of Service node. All registered QoS objects are listed under this node using the description field in the QoS object. This view enables the user to quickly browse the database for particular data-series.

The structure is organized like illustrated below:

- Quality of Service
  - Quality of Service object name
    - Source
    - Target

Note: This is the structure if the QoS objects are ordered on the QoS name. Clicking the icon beside QoS node in the Navigation Pane lets you select another sorting key, such as order by QoS group, where the QoS objects will be grouped into logical groups.

Let's explain the structure using "real" data. The cdm probe provides the QoS data registers the QoS objects it will use during the startup sequence. These messages are collected by the Data Engine and transformed into database tables and definitions. The cdm probe will, on a timed interval, send its QoS data, based on its findings and calculations to the Data Engine as messages (subject: QOS_MESSAGE).

This will populate the database tables matching the cdm QoS configuration. Let's say we configured the cdm to send QoS on CPU usage and Disk usage (C:) data. You are now able to browse the data using the Service Level Manager. Find the Quality of Service node and double-click to expand it. This brings up the QoS descriptions registered by the Data Engine. Find the Disk Usage node and double-click to expand it. The children nodes will reflect the sources (Robots) that are generating the QoS data. The target node holds the last piece of information needed to pinpoint the recorded data in the database, in this case e.g. C:. We can address each QoS using the QoS-Name, Source and Target.
Generating a Graph

The Service Level Manager generates a graph when double-clicking a QoS objects target node in the Navigation Pane.
You may select the period from the drop down list (last day, last week etc.).

Using the field Max Value, you may also set the maximum value of the vertical axis in the graph to a proper value to compress the graph in cases where you have "peaks".

**Exporting the QoS Data Series**

You may export QoS data to a file, using the following method.

Using the Export QoS Data wizard, accessible from clicking the icon beside the respective QoS in the Database Status dialog and selecting Export Data option from the menu (see Viewing the Actual Database Usage (see page 364)).

Note that this wizard may also be started, clicking the icon beside QoS constraint under the QoS node in the Navigation pane and selecting Export data.
Exporting QoS Data Using Export QoS Data Wizard

1. Select Tools > Wizards > Export QoS Data from the menu-bar to launch the wizard.
2. Click the Next button to continue.
   The next dialog lets you configure the QoS source and targets.
   (Note that the fields in this dialog are already filled in if the wizard was started from the Database Status window).
3. Click the Next button to continue.
   The next dialog lets you select which period to export.
4. Configure the period to select data from and click the Next button to continue.
5. In next dialogue box, modify the column and or row separators if wanted, and include or exclude column headers in the formatted file.
6. Click the Next button to continue to the last dialog.
7. Click the Finish button to finish and exit the wizard.

Two other ways to launch the Export Data wizard:

You may also launch the wizard by selecting Tools > Database Status from the menu-bar.

This will open the Database Status window, showing you various information related to the QoS data stored in your database.
Click the icon beside the respective QoS in the list which opens a small menu. Selecting the Export Data option will launch the wizard.

Also clicking the icon beside the respective QoS object under the Quality of Service node in the Navigation pane and selecting Export Data will launch the Export wizard.

Data Management

All QoS data is received by the data_engine and inserted into the database. The data_engine may be configured to perform automatic clean-up procedures by configuring the data management section of the data_engine.

You can also perform manual data-management in SLM, by clicking the Tools tab > Data Management.

Viewing Actual Database Usage

Select Tools > Database Status from the menu bar to bring up the Database Status window. This will show you various information related to the QoS data stored in your database. The tabs represent various ways of getting information regarding the database usage.
The Active Objects Tab

Shows all available QoS objects registered within the database structure. Bring up the action menu by issuing a right-mouse button-click. View the QoS data by double-clicking the list element, or perform a selective data-management by selecting delete.

- F5 - Refresh list
- F6 - Update/view sample period for the selected QoS objects.

The Sub menu

On clicking icon beside any of the QoS objects in the list opens a small menu with the following options:

Update

Updates the selected row with data from the current compliance period.

Get Statistics

Finds the details statistics like Historic Row, First Sample, Last Sample, S./Min etc. within the current compliance period. This information will then be found in the respective columns in the window.

Delete Object(s)

Deletes the selected QoS object(s) (not only the data series) from the database. The deleted QoS object will disappear from the database and will not be re-created before the probe is restarted.

Merge Objects

Allows you to merge two QoS objects of the same type. Left-click the first object and <ctrl>+left-click the next object. Click icon and select the merge object option.

Note that you can toggle the direction of the merge operation by clicking the arrow in the middle of the dialog. The newly merged QoS objects will take the “destination” QoS object’s name.

You also have the option to delete the “source” QoS after the merge operation. The deleted QoS object will disappear from the database and will not be re-created before the probe is restarted.

View Data

Opens the graph, displaying the data for the current compliance period.

Delete Data

Opens the Data Management dialog for the selected QoS constraint, allowing you to delete the complete QoS data series, or just for a selected period. The QoS object is not deleted selecting this option, and data for the selected QoS constraint will continue to be stored in the database table.

Export Data
Opens the Export QoS Data wizard, enabling you to export the data series for the selected QoS constraint dialog to a file. See the section Exporting the QoS Data Series (see page 363).

**Filter by QoS**

Selecting one entry in the list and selecting Filter by QoS, all entries with the same QoS as the selected one will be listed, all others will be removed from the list. In the example below, all entries with QOS_CPU_USAGE will be collected in a list; all others will be removed from the list.

Click the icon beside the respective QoS and select Update to return to the original list.

**Filter by Source**

Same as Filter by QoS, but in this case, entries with identical source will be collected in the list.

**Filter by Target**

Same as Filter by QoS, but in this case, entries with identical target will be collected in the list.

**Filter by Host**

Same as Filter by QoS, but in this case, entries with identical host will be collected in the list.

**Filter by Robot**

Same as Filter by QoS, but in this case, entries with identical robot will be collected in the list.

**Filter by Probe**

Same as Filter by QoS, but in this case, entries with identical probe will be collected in the list.

**Filter by Origin**

Same as Filter by QoS, but in this case, entries with identical origin will be collected in the list.

All messages received by a given hub are stamped with an origin element. The default origin name will be the HUB name.

**Change origin**

Selecting one or more entries in the list, clicking the icon and selecting Change Origin lets you change origin for the selected entries (all messages received by a given hub is stamped with an origin element. The default origin name will be the HUB name). Available origins will be listed in the dialog popping up. You can also create a new origin by clicking the Add button.

**The columns in the window**

The followings columns appear in the window when the Active Objects tab is selected:
Quality of Service

The name of the QoS object.

Source

From where the sample originates.

Target

The target of the sample.

Rows

The number of rows in the database table containing data for the QoS Object.

Note that this information is made available after clicking the icon beside the entry and selecting Get Statistics option.

Historic rows

The number of rows in the historic database table containing data for the QoS Object.

Note that this information is made available after clicking the icon beside the entry and selecting Get Statistics option.

First sample

The time and date of the first sample within the current compliance period.

Note that this information is made available after clicking the icon beside the entry and selecting Get Statistics option.

Last sample

The time and date of the last sample within the current compliance period.

Note that this information is made available after clicking the icon beside the entry and selecting Get Statistics option.

Table id

An ID number assigned to each of the QoS objects registered within the database structure.

S./min.

The number of samples per minute.

Note that this information is made available after clicking the icon beside the entry and selecting Get Statistics option.

Originator

The address of the computer running the probe.

Host

The IP-address of the host hosting the probe from which the sample originates.

Robot
The name of the robot from which the sample originates.

**Probe**

The name of the probe from which the sample originates.

**Origin**

This is the origin of the QoS sample. All messages received by a given hub are stamped with an origin element. The default origin name will be the HUB name.
**QoS Definition Tab**

Shows a list of QoS definitions and their properties. Allows you to create new QoS definitions, or edit or delete existing QoS definitions.

**Name**

Name of the QoS.

Click the icon next to the name to display a popup menu with these choices:

- **New**
  
  Opens the QoS Definition dialog, where you can define a new QoS.

- **Properties**
  
  Opens the QoS Definition dialog for that QoS, where you can view its properties.

- **Delete**
  
  Deletes the QoS.

- **Update**
  
  Updates the QoS definition table.

**Group**

Group the QoS belongs to. This determines how QoS are grouped in the tree in navigation pane.

**Unit**

Unit of measurement for the QoS.

**Abbreviation**

Abbreviation for the unit of measurement for the QoS. For example, if the unit is \textit{Bytes/sec} the abbreviation is \textit{B/s}. You can use either the \textit{Unit} or the \textit{Abbreviation} when creating reports.

**Is Boolean**

Whether or not the value for the QoS is Boolean.

**Has Max**

Whether or not there is a maximum value for the QoS.

**Type**

Data type of the QoS.

**Objects**

Number of objects this QoS definition is assigned to.

**Rows**

Number of rows in the database with entries for the QoS.
Using SLM

**Size (KB)**
Size, in kilobytes, of the rows with entries for the QoS.

**Historic**
Number of measurements in the historic database for the QoS.

**Size (KB)**
Size, in kilobytes, of the historic measurements for the QoS.

**The QoS Probes Tab**
Shows the QoS objects registered per probe. Delete the selected data-series by clicking icon and selecting Delete Data option. This operation deletes all objects matching the "host" field and the "QoS" field.

The followings columns appear in the window when the QoS Probes tab is selected:

**Registered**
The time the QoS object was registered in the database,

**Host**
The IP address of the host where the probe sending the QoS data is installed.

**Probe**
The name of the probe sending the QoS data.

**Quality of Service**
The name of the QoS object.

**Information Tab**
Shows information related to the database, such as:
- Database server name
- Database name
- SLM version
- Database owner
- When the database was created
- Database usage
- Database location
Managing your QoS Data

Select Tools > Data Management from the menu-bar to bring up the Data Management Dialog, where you can change the period settings and delete historic data for the selected QoS constraint. The various fields in the dialog are described below:

**Quality of Service Object**

**Description**
The QoS description string.

**Object**
The QoS object name.

**Source**
The source of the QoS data. E.g. computer, device etc.

**Target**
The target of the QoS data. E.g. the disk, network service, CPU, etc.

**Operation**

**Delete Data**
Select this option to delete the QoS data related to the source and target settings.

**Invalidate Data**
Select this option to invalidate the QoS data related to the source and target settings.

**Delete Historic data**
Select this check box to delete the historic data for this QoS constraint

**Period settings**
You can select a period, or you can select Ignore, which means that the measurement is not restricted to specific periods.
Watching SLA Calculations

Selecting Tools > SLA Calculations in the Menu bar opens the Service Level Calculations window, listing the current calculations for the SLAs.

A service level agreement (SLA) will automatically get recalculated for the current period by the sla_engine (on a schedule as specified in the sla_engine user interface), but you can manually recalculate a SLA by clicking icon beside desired SLA and selecting New option (see below), or clicking icon beside the SLA in the Navigation Pane and selecting Recalculate.

**New Job**

Using this option, you may manually recalculate the SLAs.

The Job properties dialog is opened, allowing you to select a SLA to be recalculated. The time period for the report can be specified, either as a Fixed report, where you can select one or more fixed periods, or as a Custom report, where you can select a time range. The Job start time specifies when the job calculation job should be started; the Job expiration time specifies when the job is removed from the database.

The Create historic data option lets you select the compliance data to be recorded/saved for historic reports, and the Delete report when expired will delete the report from the database when the report has expired.

SLAs are also automatically recalculated on a schedule as specified in the sla_engine user interface.

**Delete**

Deletes the calculations for the selected SLA from the list in the window.

**Update**

Updates the selected row with data from the current compliance period.

**View**

This option opens the compliance job description window for the selected SLA.

**The columns in the window**

The Service Level Calculations window contains the following columns:

**Job Id**

The job identification number assigned to a new job.

**SLA**

Shows the name of the SLA being calculated.

**Description**
Indicates if the calculation job runs automatically as scheduled, or if it was manually started by the administrator.

**Period Begin**
Displays the starting date/time of the compliance calculation period.

**Period End**
Displays the end date/time of the compliance calculation period.

**Owner**
The name of the operator that requested the calculation job

**Job Start**
Defines the date/time for when the compliance calculation job should start.

**Job Expire**
Defines the date/time for when the job is automatically removed from the SLM system.

**History**
Indicates that compliance data should be recorded/saved for historic reports.

Double-clicking one of the SLAs in the window opens the Calculated SLA Compliance job window for that SLA.

The window displays:
- The selected Compliance period
- The name and description of the SLA
- The Status, with the expected and the achieved value
- The SLOs, listed in the lower left corner
- The QoS constraints, listed in the lower right corner, supplying the following information:
  - The name of the QoS object
  - The name of the source computer
  - The name of the target computer
  - The achieved compliance
  - The expected compliance
  - The total number of samples

Double-clicking one of the QoS constraints brings up the QoS graph (also see the section [Generating a Graph](see page 362)).

**SLA ID**
The identification number assigned to the SLA.
SLM Database Structure

The QoS data structure is dynamically created by data_engine on the first startup, and when the first unique QoS definition or QoS message is received from a probe.

The tables created in the SLM database have prefixes indicating the type of data they contain.

The naming convention for the tables is as follows:
- S_ for tables used to store system data
- D_ for data tables
- H_ for tables containing historic data
- HN_ for data tables containing historic/compressed data
- RN_ for data tables containing unprocessed (raw) data directly from the probes

QoS Data Tables

The QoS data structure is dynamically created by the data_engine on the first startup, and when the first unique QOS_DEFINITION or QOS_MESSAGE message is received from a probe.

The S_QOS_DEFINITION table contains the definitions of known QoS types (e.g. QOS_CPU_USAGE), and is updated when a probe sends a QOS_DEFINITION describing a new QoS type.

The S_QOS_DATA table contains an index of all data tables for the QoS objects. When a probe sends a QOS_MESSAGE containing a QoS object that is not already defined in the S_QOS_TABLE, a new entry is added to the table and the data is inserted into the table referenced in column r_table (typically RN_QOS_DATA_nnnn) with the table_id that the new row is given when inserted into the S_QOS_DATA table.

Note: Do not drop the data tables manually. Instead delete the entry from the S_QOS_DATA table, and the tables will be dropped by a trigger. You must restart the data_engine afterwards.
SLA Configuration Tables

The Service Level Manager is used to describe the SLAs and the descriptions are stored in the following key tables:

- S_SLA_DEFINITION
- S_SLO_DEFINITION
- S_QOS_CONSTRAINTS

There are also other tables, containing data about exclude periods, operating periods, and so on. Here is a list of some of these tables.

- S_SLA_EXCLUDE_PERIODS
- S_SLO_EXCLUDE_PERIODS
- S_SLA_NOTES
- S_SLO_NOTES
- S_SLA_CALCULATION
- S_SLO_CALCULATION
- S_QOS_CALCULATION
- S_OPERATING_PERIOD
- S_TIME_SPECIFICATION

SLA Calculation Tables

The first step for a SLA to be calculated is that a job is added to the D_SLA_JOBS table manually by a user or automatically by the sla_engine. The sla_engine detects that there is a new calculation job and performs the calculations. The results are stored in these tables:

- D_SLA_COMPLIANCE
- D_SLO_COMPLIANCE
- D_QOS_COMPLIANCE

When the last job for a SLA in the compliance period is calculated, the result will be added to the historic tables:

- H_SLA_COMPLIANCE
- H_SLO_COMPLIANCE
- H_QOS_COMPLIANCE
Account Tables

The following tables support the Account views and data ownership functionality.

- Accounts Table (CM_ACCOUNT)
- Contacts Table (CM_CONTACT)
- Account-SLA reference Table (CM_ACCOUNT_SLA)
- Origins Table (CM_ACCOUNT_OWNERSHIP)
- Roles Table (CM_ROLE)
- Access Control Lists Table (CM_ACCOUNT_ACLS)
- Dashboard Table (CM_ACCOUNT_DASHBOARD)
- Group Table (CM_ACCOUNT_GROUP)
- Properties Table (CM_ACCOUNT_PROPERTIES)
Sending SQL Queries to the Database

The SLM portlet includes a SQL Query tool you can use to enter custom SQL queries. From the SLM portlet, select the Tools tab > SQL Query to access this tool. This function can be used to query the database directly, and is useful for testing and verification purposes.

**Important!** The SQL Query tool in SLM is intended for experienced users only.

Here you can select a query from the combo box, and send the query to the database by clicking the Start Query button.

You can do the following with the SQL Query tool:

- Use any SQL command.
- Enter carriage returns.
- Enter queries that consist of multiple statements.

**Note:** To use multiple statements, you must separate each statement with a semicolon.

- Run queries of multiple statements and a query selected from the combo box simultaneously.
- Enter multiple statements, then highlight one or more of the statements to run just the highlighted statements.
- Stop a query that is running by clicking the Stop Query button.

The result of a query will appear in a table under the Recordset tab. When you run a query that consists of multiple statements, a separate Recordset tab appears for each statement. The number of rows found is also displayed on the tab header.
Size Limits for SQL Queries

In environments with extensive monitoring, database tables may contain a considerable number of rows. In this situation with a previous UMP version, a SQL query in SLM could cause UMP to become unresponsive. For this reason, SLM SQL queries in UMP 6.0 are now limited to a maximum size of 1000 rows; columns are now limited to a maximum of 1024 bytes.

Although it is not recommended, you can override these limits. To do so, edit the <webapp/slm> section of the wasp.cfg file, and add the parameters max_rows and max_field. See the following example.

Example

<slm>
    reloadable = true
    cross_context = true
    load_on_startup = true
    unpack_war = true
    path = /slm
    max_rows = 10
    max_field = 2048
</slm>
The Service Level Management application window consists of the following main parts:

- The Menu Bar
- The Navigation Pane
- The Main Window Pane
The Menu Bar

This section gives a short description of the different functions and tools found in the menu line. Note that some of the menus are partly restricted, depending if your user is classified as operator or super user.

New

This option can be used when defining a new:

Service Level Agreement

Opens the Service Level Agreement dialog, where you can create new SLAs. See description in the section Creating Service Level Agreements (see page 312).

Operating Period

Opens the Operating Period dialog; see description in the section Creating an Operating Period.

Quality of Service

Opens the QoS Definition dialog; see description in the section Defining a Quality of Service object.

Tools

This option can be used when defining a new:

Database Status

Opens the Database Status window. The window contains relevant database information, such as:

- Database information, such as server name and database name, size and location.
- Listing QoS objects and probes enabled for sending QoS messages to the database.
- See description in the section Viewing the Actual Database Usage (see page 364).

Data Management

Opens the Data Management dialog. The data_engine may be configured to perform automatic clean-up procedures, but using the Data Management dialog, you may perform manual data-management.

SLA Calculations

Opens the Service Level Calculations window, listing the last calculations for currently running SLAs. Double-clicking the entries in the list, you can observe the calculated SLA Compliance for each of the SLAs in separate windows.
QoS Monitors

Selecting this option, the QoS Monitors window will be launched, listing the QoS Monitors defined (if any).

QoS Monitors can be defined by clicking icon beside a QoS object in the Navigation pane, selecting Create Monitor.

This feature will be activated and visible, if the qos_engine probe is installed.

View SLM Alarms

Displays alarms related to Service Level Management in the Alarm Console portlet in a new window.

Wizards

Lets you start a wizard for creating one or more SLAs, based on your selections through the wizard. A group will be created with one or more SLAs in the group. There are different types of wizards:

- Creating SLAs from scratch, based on a selected service, see Example — Creating a SLA based on a service (see page 348).
- Creating SLAs based on an existing SLA. Selecting a SLA in the Navigation Pane, the wizard will create a new SLA, based on the properties for the selected SLA.
- Exporting QoS data. For more details, see Exporting QoS Data Using Export QoS Data Wizard (see page 364).
- Adding an Excluded period. Normally this is done individually for SLAs/SLOs in the respective dialogs. Using this wizard, you can add an Excluded period and attach the period to some or all of your existing SLAs/SLOs.

SQL Query

Opens the SQL Query dialog, allowing you to send SQL queries to the database. This is useful if you want to test and view database contents. Also, see Sending SQL Queries to the Database (see page 377).

Window

Open Windows

Shows the list of currently open windows. At any given time, you can open any number of windows.

Maximized

If this option is selected, by default the window opens in its maximized state.
**Save**

Lets you save new definitions, or any modifications to existing definitions.

**Help**

Provides version information and shortcuts to online documentation.

---

**Preferences Dialog**

Access the Preferences dialog by clicking the **Settings** icon (Settings), then **Preferences**.

The following options and properties are available:

**Height**

Enter the height, in pixels, for this instance of the portlet to open to.

**Maximized Height**

Enter the height, in pixels, for this instance of the portlet to display at when the maximize icon (Maximize) is clicked.

**Edit QoS Constraint on double-click**

When this option is checked, double-clicking a QoS constraint in an SLO dialog opens the properties dialog for the QoS constraint. When this option is not checked, double-clicking a QoS constraint in an SLO dialog opens the QoS constraint graph.

**QoS Data Folders**

This option lets you organize the QoS objects located under the QoS node in the Navigation Pane into folders.

**Minimum number of objects to create a folder**

Lets you define the minimum number of objects that must be present for objects to be automatically organized into folders.

**Create folders for network addresses**

Checking this option enables automatic folder creation, based on network address, if the minimum number of objects you defined is reached.

**Create folders for host names**

Checking this option enables automatic folder creation, based on host name, if the minimum number of objects you defined is reached.

**Order by QoS Group, Order by QoS Name, Order by Probe**

Choose up to six data selectors under each heading to change the hierarchy of the tree in the navigation pane. The available data selectors are ACCOUNT, ORIGIN, HOST, ROBOT, PROBES, SOURCE, QOSNAME, QOSGROUP, TARGET, and NONE.

**Note:** The data selectors are not combined in any way.
Save

Allows you to save any modifications to your preferences.

The Navigation Pane

The Navigation Pane is present to the left-hand side of the application. It can be collapsed by clicking icon and can be opened again by clicking icon. It contains a tree-like structure, showing the various elements/nodes in the Nimsoft Service Level Management product suite.
Service Level Agreements

This node lists all defined SLAs. The SLAs may appear as:

- Single SLAs
- Grouped SLAs - The SLAs may be placed in one group, or in sub-groups on multiple levels under the main group. SLAs can be moved between groups, using drag-and-drop.
- SLAs attached to Accounts - Under the Accounts node, you will find all Accounts available. When creating SLAs, these can be attached to one of these Accounts. Under these Accounts, you may again create groups where you place the SLAs.

Click the icon to define new SLAs or SLA groups.

**New SLA**

Opens an empty SLA dialog, where you can define a new SLA (see the section Creating Service Level Agreements (see page 312)).

**New Group**

Opens a New Group dialog, where you can define a new SLA group. Give the new group a name and optionally a description.

**Refresh**

Updates the Service Level Manager to reflect the latest modifications.

An SLA group or an Account is selected:
View

Opens the current group in edit mode. You can view or edit the desired group details and save it.

New Group

Opens a New Group dialog, where can define a new SLA group. Give the new group a name and optionally a description.

New SLA

Opens an empty SLA dialog, where can define a new SLA (see the section Creating Service Level Agreements (see page 312)).

Delete

Deletes the selected SLA group, including all SLAs in that group.

Recalculate

Starts a new calculation job for all SLAs in the selected group.

Note: Clicking Recalculate option of a SLA group will not open the Job Properties dialogue. To see the Job Properties dialogue, select the Recalculate option of individual SLA within the group.

Refresh

Updates the Service Level Manager to reflect the latest modifications.

An SLA is selected:
View
Open the current SLA in edit mode. You can view or edit the desired SLA details and save it.

New
Opens an empty SLA dialog, where you can define a new SLA (see the section Creating Service Level Agreements (see page 312)).

New Clone
Makes a copy of the selected SLA, and appends the name of the clone with the text (COPY X). Note that if you clone an SLA that has a name that is longer than 57 characters, the name of the clone is truncated.

New Based On
Starts the SLA wizard, enabling you to create a new SLA, based on the selected one. Note that some input fields are not editable.

Delete
Deletes the selected SLA.

Recalculate
Starts a new calculation job for the selected SLA. The Job Properties dialog is opened, allowing you to edit the job parameters.

Refresh
Updates the Service Level Manager to reflect the latest modifications.
Operating Periods

This node lists the defined operating periods. Operating Periods are used to constrain the measured QoS values to specific hours of any day (the period the QoS constraint is valid.)

The operating period is defined to be a collection of time periods, and is used when we define new SLAs. We use the Operating Periods while defining the QoS constraints (see the section Creating an Operating Period)

This means that samples falling outside these time specifications will not influence the SLO/SLA compliance requirements.

Click 📜 icon and select New Operating Period which opens an empty dialog, enabling you to define a new Operating Period (see the section Create an Operating Period).

Click 📜 icon that is beside one of the defined Operating Periods listed under the node, which opens a small menu containing four options:

- **View**
  
  Opens the existing Operating Period in edit mode. You can view or edit the existing Operating Period and save it.

- **New**
  
  Opens an empty dialog, enabling you to define a new Operating Period.

- **Refresh**
  
  Updates the Service Level Manager to reflect the latest modifications.

- **Delete**
  
  Deletes the selected Operating Period.
Calculation Profiles

This option allows you to define your own calculation profiles.

These calculation profiles are used when defining the calculation properties for Service Level Objects and Quality of Service Constraints.

When defining calculation profiles, the profiles will be grouped either as SLO calculations or as QoS calculations, depending on if the selected plug-in supports single-data or multi-data series. See the section Creating a Calculation Profile (see page 336) and Compliance Calculation.

Click the icon beside the SLO Calculation sub-node (or one of the defined SLO calculation profiles), selecting New, opens an empty dialog enabling you to define a new SLO Calculation profile.

Click the icon beside the QoS Calculation sub-node (or one of the defined QoS calculation profiles), selecting New, opens an empty dialog enabling you to define a new QoS Calculation profile.

Clicking icon offers four other options, common for both sub-nodes.

**View**

Opens the existing Operating Period in edit mode. You can view or edit the existing Operating Period and save it.

**New**

Opens an empty dialog, enabling you to define a new Operating Period.

**Refresh**

Updates the Service Level Manager to reflect the latest modifications.

**Delete**

Deletes the selected Operating Period.
Quality of Service

Under this node, you will find the available registered QoS objects, grouped into logical groups based on the description field in the QoS object. This view enables the user to quickly browse the database for particular data-series.

Click icon beside QoS node that opens a small menu containing four options:

**Browse Menu Editor**

Launches the Browse Menu Editor, enabling you to define your own ways to group and sort the QoS objects listed under the QoS Node.

Choosing **New**, a new sorting view will be added to the list. The new entry will by default be given the name New Menu Item. Rename it, giving it a descriptive name.

Select your sorting criteria, one or more, from the Data Selector drop-down menus and select descending or ascending sorting order.

Click the OK button, and you will be asked if you want to save the view.

Choosing **Delete**, the selected view will be deleted.

When saved, the entries in the list will appear as menu items in the menu available when you click icon beside the QoS node in the Navigation Pane.

They will be presented in the order they appear in the list in the Browse Menu Editor.

When saved, the new views will appear as menu items when you click icon beside the QoS node in the Navigation Pane.

**Order by QoS group**

All QoS objects will be presented in their logical groups.

In addition, see the section Properties for information on sorting and organizing QoS objects in folders.

**Order by QoS**

All QoS objects will be listed alphabetically (and not grouped).

In addition, see the section Properties for information on sorting and organizing QoS objects in folders.

**Order by Probes**

All the QoS objects will be listed as per the Probes.

Main Window Pane

This is the application frame. All child windows are contained within the frame of the main-window.
SLM Terms and Definitions

This section contains definitions of terms related to the SLM portlet.

Automatic (Interval) data type

QoS Data is recorded at intervals, individually specified in the probe configuration for each of the probes.

Asynchronous data type

QoS Data is recorded each time the measured value changes.

Calculation method

Calculation method is the set of rules and conditions determining the way the SLA compliance is calculated.

Compliance period

The compliance period defines the time interval--day, week or month--used for calculating an SLA’s compliance.

The following illustration shows the relationship between the operating period and the compliance period.

Example
In the above example, the gray area is the compliance period, defined as one week (Monday to Monday). The blue area is an operating period between 08:00 and 17:00 every weekday, and includes five time-specifications. The blue plotted line within the time-specifications of the operating period is used in compliance computations for the SLA.

**Compliance Percentage**

A compliance percentage is the percentage of time a QoS constraint is within the thresholds of a defined time period.

The sla_engine probe checks each sample taken in a compliance period. The result is compared and summarized as failed or successful.

In the illustration above, the red line represents the threshold value, the green line represents the average value and the blue line represents the actual sample values.

None of the samples breach the threshold line within the operating periods, which means compliance is 100 percent. The five samples that breach the threshold are outside of the compliance period, which was Monday to Monday, with operating periods every weekday from 08:00 to 17:00.

If, for example, the total number of samples within the operating period were 129, and nine samples were breaching the threshold, 6.98 percent of the samples would be considered out of compliance (9 * 100/129). If this were the only data defined in the SLA, and the SLA required 98.50 percent or better compliance, the SLA would be breached due to a compliance percentage of 93.02 percent (100 percent - 6.98 percent).
Data Types

We use the following different data types when calculating the compliance:

Automatic (Interval)
QoS Data is recorded at intervals, individually specified in the probe configuration for each of the probes.

Asynchronous
QoS Data is recorded each time the measured value changes.

Domain
The Domain is the top-level node in the hierarchy, and a site is normally set up with one Domain. It is used to group one or more Hubs in a logical set such as a company or enterprise.

Hub
The Hub is a message concentrator and re-distributor. It is the collection point for all messages coming from the various installed Robots. Many other Nimsoft components can connect to the Hub to receive dedicated messages and perform other specific activities. One such component is the Alarm server.

NULL value
A NULL value will be recorded into the database typically if a probe does not measure a value from the target due to a timeout, e.g. no answer to a ping). In the QoS Calculation Profiles, available under the Calculation Profiles Node in the Navigation Pane of the SLM, you may define how these NULL values are handled in a compliance calculation.

Operating period
The operating period constrains the QoS samples to one or more time-specifications within the compliance period. This means that samples falling outside these time specifications will not influence the SLO/SLA compliance requirements. Each operating period is defined as a union of one or more time-specifications.
**Probes**

Probes are small dedicated pieces of software that monitor specific resources or events. Each probe can be easily configured for your own specific monitoring requirements.

**Robot**

The Robot is the first line of management for the Probes. The Robot starts and stops the probes at the required times, collects, queues and forwards messages from the probes onto the specified Hub.

**Service Level Agreement (SLA)**

A Service Level Agreement (SLA) is an agreement to deliver a service within a specified/fixed time-period to an extent where both parties agree on a measurable service levels. The parties may be an IT department delivering services to another department within the company, or by a company and an external service provider.

The services included in the SLA may be a collection of monitored objectives we call Service Level Objectives (SLO). These objectives (or group of objectives) are monitored by dedicated programs (often standard probes) that monitor e.g. network connectivity, application (Oracle, Exchange, e-mail) availability and service (DNS, DHCP) availability.

**SLO**

A Service Level Objective (SLO) is a combination of one or more component measurements (Quality of Service) to which constraints are applied. A SLO is said to be in compliance if the underlying measurement values are within the specified constraints. SLO’s may have operating periods during which the SLO has to be compliant.

**SLM**

The Service Level Manager (SLM) is the application where service level configuration and monitoring is performed. The application needs a valid license to operate.
**QoS**

The Quality of Service (QoS) is the atom of the Service Level Management. The QoS is the actual value (sample) collected and used centrally to determine the state of the service level objective. This value is normally collected by a probe like cdm, net_connect, url_response etc. The value is first used for alarm purposes, but if the probe is configured to deliver Quality of Service, then a QoS message is issued.

A QoS constrained by threshold, source, target and operating period settings is used as the building blocks for SLO’s.

**QoS object**

The QoS object is defined by its QoS name, source and target. All sample-data with this unique combination form data-series that may be used as part of any Service Level Objective.

**QoS constraint**

A constrained QoS object is defined by its QoS name, source, target, threshold and operating period. This constrained object is the building block of the Service Level Objective.

**Troubleshooting SLM**

**SLM May Allow Invalid Logical Expressions**

To use logical expressions for calculation profiles in the SLM portlet, you must enclose the expressions in parenthesis to be valid. If you don’t enclose a logical expression in parenthesis, SLM will allow you to save it without displaying a warning message that the expression is invalid.
The SLM Portlet Does Not Record a QoS Object

**Symptom**
The probe has been configured, but the QoS object does not appear in SLM.

**Solution**
This may occur if you configure a probe to deliver QoS before you install the data_engine. In this case, the QOS_DEFINITION message is ignored, and the QoS object does not appear in SLM.

Restart the probe (deactivate/activate) or restart the robot if you have modified the configuration of more than one probe.

If you restart the probe or robot and the QoS object still does not appear under the QoS node, wait a moment, and then select **Update** from the action menu.

SLM Portlet Displays Incorrect Data Set

Depending on factors such as the user’s time zone, the time zone of the database, and whether or not SLAs have been assigned a time zone, users may see the wrong data set in the SLM portlet. If this occurs, switch to the SLA Reports portlet to view SLA data.
Chapter 24: Unified Reporter

Unified Reporter (UR) is an optional component that provides advanced reporting for the Unified Management Portal (UMP).

UR provides the following features:

- Drag and drop ad hoc report building.
- Drag and drop dashboard building, with live refresh, and mashups of external content.
- Built-in charting that includes pie, bar, line, multi-series, area, and many other chart types.
- A library of out-of-box (OOB) reports that save time and effort.
- Database abstraction using JSON query extractors.
- Self-service parameterized web reporting.
- Report scheduling, distribution, and historical versioning.
- Access to any data source, including the Nimsoft SLM database.

In addition, UR provides developers and power users with:

- The iReport graphical report designer for building more complex reports.
- Print-ready pixel-perfect production reporting.

To obtain documentation for Unified Reporter, visit the Unified Management Portal web-based help.
Chapter 25: Unified Service Manager

The Unified Service Manager (USM) portlet offers an easy way for you to visualize and organize your infrastructure and to configure monitoring in CA Nimsoft Monitor.

USM’s dynamic and flexible grouping features allow you to organize your infrastructure into an unlimited hierarchy of groups and subgroups. You can organize groups according to concepts such as service, customer, geography, organization, technology, etc. Membership in groups is managed automatically based on rules, defined by you, that filter on device attributes. Servers and network devices are automatically added to groups whose attributes they match. Once your rules are defined, USM manages and updates group membership with newly discovered devices.

Groups allow you to apply a set of monitoring parameters to multiple systems from a service-oriented perspective. This is called service-oriented configuration (SOC). SOC provides simplified configuration of monitoring through the use of policies, called monitoring templates, that hide the underlying traditional CA Nimsoft Monitor individual probe configuration and deployment. You create monitoring templates by assigning one or more monitors. Monitors are metrics such as CPU usage, ping response time, or URL monitoring. Monitoring templates allow you to configure a monitor once and use it multiple times across your environment. You can configure monitor settings such as thresholds and data collection intervals. When a monitor is assigned to a group, probes are automatically pushed to robots and configured with the appropriate settings for each monitor.
USM also has these features:

Executive views

Out-of-the-box executive-level views, or badges, show the overall status of critical services and customers. You can drill down into each badge to view detailed availability, performance, and alarm data for groups and devices, and can switch in context between three levels of badge views and a hierarchical tree view.

Integrated alarm view

An integrated alarm console allows you to view alarms for each group or device. You can move between summary and detailed views, and can view alarm details and history in-line in the console. Also available in-line in the console are performance graphs for the metric that triggered the alarm, with the alarm occurrence indicated on the graph.

Multi-tenancy support

Portal administrators can define which accounts have access to groups so that each account is able to view only its own groups.

Geo View

Easily create hierarchical visual representations of monitored groups and members. You can place drilldown views within geo views to provide additional details.

Discovery of your IT asset inventory

Discover devices on your network using the Discovery Wizard or by importing devices from an XML file. Easily view the devices discovered for each network range you defined.

Device views

Out-of-the-box views provide dynamic information about the device, including performance graphs and a high-level view of alarms, disk usage, and interface status. You can drill down to view alarm details or performance graphs for disk usage and interface traffic.

Out-of-the-box device reports

Out-of-the-box device dashboards and reports provide consolidated views of QoS, availability, and monitoring, and continue the move away from legacy components such as the dashboard_engine probe and the Dynamic Views portlet.

Metrics

Performance data is displayed for each device. The metrics displayed vary according to the type of device and the probes monitoring the device, and can include, for example, metrics related to system uptime, database server performance, or memory usage on virtual machine hosts. Alarms related to a metric are indicated on the graph.
Minimum, Maximum, and Trending Data

In addition to showing the average values of the intervalized data, the graphs in USM display the minimum and maximum values for each interval. The minimum and maximum values are displayed as shaded areas below and above each point in the graph. The data graphs also display a linear trend-line indicating whether the values in the chart are increasing or decreasing. These graph features are also available when you click the Pop out icon and launch the PRD graph in a new window.

Customizable report templates per group

Create custom reports using the Performance Reports Designer and use the reports as templates for groups. Multiple report templates can be assigned to groups so that you have multiple report choices when viewing device details. Select a report type to view real-time performance reports for a device.

Auto robot deployment

The auto robot deployment feature allows you to distribute robots to systems in bulk. You can specify the target systems in USM or import a list of systems from an XML file. USM automatically installs, configures, and launches robots on the target systems.

Automatic grouping

Dynamic groups can be created according to several criteria. The groups are created under a parent group that you can name. For example, an MSP can automatically create groups by Origin under a parent group named Customers. This automatically creates a dynamic group in USM for each MSP customer.

Enforcement of monitoring policies

Monitoring policies are enforced through the inheritance of monitoring templates. All members of a group, including subgroup members, inherit monitoring templates. This allows you to enforce a minimum standard set of monitors for all existing and future devices in any subgroup. Group priorities can be configured to override lower level templates to enforce top-level monitoring policies. Or, you can set lower level groups with a higher priority to override the inherited policy.

Standalone USM

A standalone version of USM (outside of UMP) can be launched using a URL or an HTML file. This allows you to link to USM from other applications, or to integrate data for a group or system in USM into a web page. For example, if you are concerned about a particular service or group of customers, you can display a view of their availability, performance, or alarm data in a frame on your home page.

This section contains the following topics:

Access to USM (see page 402)
Navigating in USM (see page 402)
Geo View (see page 410)
Access to USM is controlled by the NMS Access Control List (ACL). To view USM, a user must have the **USM Basic** permission set in the ACL. Otherwise, a "Permission Denied" message is displayed when the user tries to run USM. Additional ACL permissions control whether a user can modify groups, monitors, and monitoring templates; deploy robots automatically; or take actions on alarms. For a complete list of permissions for USM, see [Set Permissions for UMP Portlets](see page 504).

The views and tabs that a user can see in USM are controlled by the UMP Views settings in the Account Admin portlet. See the sections [Edit ACLs Dialog](see page 27) and [Configure UMP Views Dialog](see page 29) for more information.

Navigating in USM

USM provides several tools to display and find information. You can view your groups and systems in different types of views, the tree view or several badge views, and you can search for systems using either a basic or advanced search. If Relationship Viewer is running, you can launch Relationship Viewer in context, displaying relationships for the group or system selected in USM.

Views

USM provides several ways to view the systems in your environment. You can choose the tree view or one of three badge views. The tree view allows you to access detailed information, while the badge views provide high-level information at a glance.

Choose the view by clicking one of the icons in the toolbar.
**Tree View**

The tree view displays your groups in a hierarchical tree in the navigation pane to the left. A status icon indicates the highest severity alarm for each node in the tree, and the number of systems in each group is indicated in parentheses after the group name. As you hover over a group name, icons appear that allow you to add, edit, or delete groups.

The pane on the right displays detailed information about groups, alarms, or systems.

Click on a group name in the tree to see a table listing information about group members. Click a column header in the table to sort by that column. Click the **Alarms** tab to see information about alarms for the group.

Click the name of a system in the tree to view system properties and performance data for the past 24 hours. Click an alarm icon to view alarm data for the system. If the system reports disk usage information, click the usage bar graph to view a Performance Report of disk usage data.

**Badge View**

The badge views display groups or systems as icons. There are three different badge views at different "zoom" levels. The functionality is the same in all three views, but the amount of information displayed on the badges varies.

All of the badges have icons that indicate alarm status and OS type, and the color of the badge indicates alarm status. The Badge 2 (medium badges) view also displays the system name. The Badge 1 (large badges) view also displays the number of alarms and, for groups, the number of elements in the group and all its subgroups. In the Badge 2 and Badge 3 views (medium and small badges), hover over a badge to see all the information for that badge.

In all three badge views, you can drill down to the next level by clicking on a badge. Click on a badge for a system to view detailed information about the system. This is the same system information displayed when you click on the name of a system in the tree. To move back up a level in a badge view, click the bread crumb link.

**Paging**

If there are more than 100 items in a group, the first 100 are shown and you can page through the remaining items.

To page through items, click the View Next or View Previous icons. These icons are displayed only if there are more than 100 subgroups or systems in the selected group. The View Previous icon is displayed once you page past the first page.
In the badge views, the paging icons are under the toolbar in the top left of the window.

In the tree view, the paging icons are displayed next to the name of the group when you mouse over the group name.

You can also page through items in the Members table in the tree view.
Sorting

You can sort groups or computer systems in the badge or tree views according to:
- Name
- Severity
- Alarm Count
- Element Count

To sort items, choose the sorting criterion you want from the Sort by pull-down menu.

USM Alarm and Device Icons

Icons indicate alarm severity and device type in USM. These icons appear in the badge views, tree view, and device view.

The following icons indicate the highest severity alarm for the system:

- Critical
- Major
- Minor
- Warning
- Informational
- Clear
The following icons indicate the type of device:

- Group container
- Group
- Hub
- Linux
- Robot
- Router
- Server
- Solaris
- Switch
- Unix
- Unknown device type
- Virtual server
- VMware
- Windows
Search for Computer Systems

You can search for computer systems according to various criteria. For example, you can search for all systems with a certain OS Type or with a certain Origin.

There are two types of search, Basic Search and Advanced Search. Both types search through all of your systems, regardless of whether you have selected a group in the tree or badge views.

**Basic Search**

Use to do a full-text search for any of the fields in the Members table, except for Nimsoft Type and State.

**Advanced Search**

Use to search for Nimsoft Type or State, to search for multiple criteria, or to filter out certain criteria.

**Note:** If you search on origin, both Basic Search and Advanced Search will only match systems using the discovered origin, the first origin listed in the device details. Any enriched origins listed below the discovered origin are not used.

**Basic Search**

You can use Basic Search to do a full-text search for any of the fields in the Members table, *except* for Nimsoft Type and State.

**Note:** You cannot search for Nimsoft Type (robot, hub, etc.) or State (managed, unmanaged, etc.) using Basic Search. To search for these attributes, use Advanced Search.

**Note:** The special characters \, %, and _ may need to be preceded by or enclosed in escape characters when you use basic search or advanced search, or when you apply filters in USM or in PRD. In the case of advanced search and filters, this applies when the contains, starts with, or ends with operator is selected. The escape character required depends on the type of database. For MySQL, the backslash (\) is the escape character; for MS SQL, special characters are enclosed in square brackets ([ ]). Searching for special characters is not supported for Oracle.

The following table shows how to enter special characters for each database type.

<table>
<thead>
<tr>
<th>Character</th>
<th>MySQL</th>
<th>Oracle</th>
<th>MS SQL</th>
</tr>
</thead>
<tbody>
<tr>
<td>\</td>
<td>\</td>
<td>Not applicable</td>
<td>\ (No escape character required)</td>
</tr>
<tr>
<td>%</td>
<td>%</td>
<td>Not applicable</td>
<td>[%]</td>
</tr>
<tr>
<td>_ (underscore)</td>
<td>_</td>
<td>Not applicable</td>
<td>[_]</td>
</tr>
</tbody>
</table>
Navigating in USM

Follow these steps:

1. Enter text in the Search field, then press Enter.
   
   The text field is not case-sensitive.

   In the badge views, the search results are displayed as a list of badges. In the tree view, the search results are displayed in the Search Results Members table, and the under the Search Results node of the tree.

2. To drill down to view information about a system in the search results, click the name of the system (the badge, the node in the tree, or the row in the Search Results Members table).

3. To return to the search results, click the Search Results node in the tree (tree view) or the Search Results bread crumb link (badge view).

Advanced Search

You can use Advanced Search to search for any field in the Members table (including Nimsoft Type or State), to search for multiple criteria, or to filter out certain criteria.

Note: For Oracle, systems with null database values for a search criterion are not included in the results unless explicitly specified. For example, if you search for not OS Type is windows, systems that have null values for OS Type are not included in the search results. To include these, add or OS Type is null as a second search criterion.

Note: The special characters \, %, and _ may need to be preceded by or enclosed in escape characters when you use basic search or advanced search, or when you apply filters in USM or in PRD. In the case of advanced search and filters, this applies when the contains, starts with, or ends with operator is selected. The escape character required depends on the type of database. For MySQL, the backslash (\) is the escape character; for MS SQL special characters are enclosed in square brackets ([ ]). Searching for special characters is not supported for Oracle.

The following table shows how to enter special characters for each database type.

<table>
<thead>
<tr>
<th>Character</th>
<th>MySQL</th>
<th>Oracle</th>
<th>MS SQL</th>
</tr>
</thead>
<tbody>
<tr>
<td>\</td>
<td>\</td>
<td>Not applicable</td>
<td>\ (No escape character required)</td>
</tr>
<tr>
<td>%</td>
<td>%</td>
<td>Not applicable</td>
<td>[%]</td>
</tr>
<tr>
<td>_ (underscore)</td>
<td>_</td>
<td>Not applicable</td>
<td>[]</td>
</tr>
</tbody>
</table>

Follow these steps:

1. Choose Advanced Search from the Actions menu.

   The search filters are displayed in the tree view.

   Note: If you choose Advanced Search while in a badge view, the display switches to the tree view, where you can perform the search.
2. Choose items from the pull-down menus and enter text in the text field as appropriate.

   The text fields are not case-sensitive.

   The search filter fields work the same way as the filter fields for creating groups. For an example of how to use the filter fields to create a group, see Filter Example. (see page 450)

3. Click the Add Filter icon (○) to add additional rows to the filter if wanted.

4. Click Search.

   The search results are displayed in the Search Results Members table.

5. (Optional) To drill down to view information about a system in the search results, click the name of the system in the tree or click the row in the Search Results Members table.

6. (Optional) To return to the search results, click the Search Results node in the tree.

7. (Optional) To view the search results in a badge view, click one of the badge view icons.

**Export Search Results to CSV**

Both Basic and Advanced Search allow you to export search results to a CSV file.

After searching for systems, click the Actions menu > Export Group. You will be prompted to enter a name for the file and a location on your local system to save the file to.

**Pop Out Graphs**

Data graphs are displayed in a number of places in USM. Click the Pop out icon (○) in the upper right corner of a graph, visible when you mouse over the area, to display the graph in Performance Reports Designer in a new window. From here you can use the full functionality of Performance Reports Designer, such as changing the time interval, changing the style of the graph, or exporting the data to a .csv file.

**Launch Relationship Viewer**

If the Relationship Viewer portlet is available, you can launch the Relationship Viewer in context from USM by choosing Launch Relationship Viewer from the Actions menu.

The Relationship Viewer opens in a new browser window. If you first select a group or specific managed element, the Relationship Viewer opens with the selected element(s) centered and with the radius slider visible. If you do not make such a selection first, the Relationship Viewer opens with all managed elements visible at the lowest zoom level.
If multiple relationships are available, a sub-menu of your choices is displayed when you click **Launch Relationship Viewer**; click your preference to open the Relationship Viewer using that relationship.

**Geo View**

You can use the Geo View node in USM to create hierarchical visual representations of monitored groups and members. Driltdown views can be embedded within a geo view to provide additional details.

In addition, the Geo View node provides the following capabilities:

- You can use a custom label and image for each geo view and drilldown view (supports .jpg, .jpeg, .png, and .gif).
- You can drag groups or members onto a geo view, and position them as needed.
- The highest severity-level of an object in a geo view or drilldown view is propagated to the top level.
- The USM tree hierarchy is automatically updated to display sub-nodes for the geo views and drilldown views you create.
Creating a Geo View

Note: While in edit mode, you cannot perform most other actions in USM. However, in edit mode, you can still browse the USM navigation tree, or use the Basic search.

Note: An NMS user cannot create a geo view for an account until an account contact user first logs in and creates a geo view.

Follow these steps:

1. Hover the cursor over the Geo View node, and click the plus action item (+) that appears. Enter a name for the view.
   Note: The name field has a 255-character limit, and does not allow you to enter unsupported characters.

2. Locate the new geo view under the Geo View node, and click the gear action item that appears.

3. Click the Upload new geo view icon, and browse to an image to use for the background of the geo view.

4. Drag groups or members from the USM navigation tree onto the geo view. You can use the Basic search to find devices that are not members of a group.
   Note: You must be in edit mode to drag items to a geo view. While in edit mode, you can remove items from the geo view by hovering the cursor over the item, and clicking the delete action item (x).

5. To add a drilldown view to the geo view:

   a. While in edit mode, click the Add drilldown view icon.
   b. Position the drilldown view on the geo view. You will be able to reposition the drilldown view as needed later.
   c. Provide a name for the drilldown view.
      The drilldown view is added to the navigation tree under the sub-node of the geo view.
   d. Locate the new drilldown view under the geo view sub-node, and click the gear action item to upload an image or make other changes to the drilldown view.

6. Click the save icon to save your changes and exit edit mode.
Discovery

CA Nimsoft administrators can use the Inventory node in USM to perform discovery of hosts and devices on their network, either using the Discovery Wizard or importing host and device information from a file.

The Discovery Wizard leads you through the process of configuring a scan of the network. At a high level, first you create authentication profiles that the discovery agent will use to access systems on your network. Then you create network ranges to define which parts of your network to include in the discovery. Finally, you launch discovery and/or schedule discovery to run in the future.

To run the Discovery Wizard, the discovery_agent and discovery_server probes must be deployed and running in the domain. These probes are deployed and activated on the primary hub when CA Nimsoft Monitor is installed. However, if you have an extensive network you may want to perform distributed discovery using multiple discovery_agent probes. For more information about this advanced discovery configuration, see the Discovery User Guide.

You can also import device information from a file. To do this, you create an XML file that contains information about devices on your network. Then you import the file. CA Nimsoft Monitor populates your Nimsoft database with the information from the file.

The Inventory node in USM also allows you to view discovered systems and properties of those systems. You can view all discovered systems in your network, or only those associated with a selected discovery agent or range.

Running the Discovery Wizard

Launch the Discovery Wizard

The first time you open the Unified Management Portal (UMP) it opens to the Unified Service Manager portlet and the Discovery Wizard is automatically launched.

After the first time you open UMP, you can launch the Discovery Wizard manually if you want to run discovery or change your discovery settings. You can launch the Discovery Wizard from the Inventory node or from the Actions menu.

Note: The Discovery Wizard will not run after an update of CA Nimsoft Monitor if there are existing ranges that define excluded IP addresses. You must either choose to accept the system prompt to delete excluded ranges, or remove them manually from the database before discovery will run.
Follow these steps:

1. Hover the cursor over or click the name of a discovery agent or range in the tree. Discovery agents are indicated by the magnifying glass icon (🔍), and ranges are indicated by the network icon (🌐).

2. Click the gear icon (⚙️) to the right of the discovery agent or range name in the tree, or choose Discovery Wizard from the Actions menu.
   
   Note: The Discovery Wizard menu option is enabled only when you click on a discovery agent or range in the tree.

Navigating in the Discovery Wizard

There are a few things to be aware of when using the Discovery Wizard:

- If you click the Close button or the X icon in the title bar before completing the Discovery Wizard, you see a prompt asking whether you want to save your changes. If you execute discovery by clicking Finish on the final screen of the Discovery Wizard, changes are retained.

- If valid information is entered in the required fields of an authentication profile or network range, the information is automatically saved when you click Next. Required fields are outlined in red.

- Passwords for authentication profiles are displayed as asterisks. If you want to see a password as you enter it, click the Show password icon (👁️) next to the Password field. After you click Next, the password is displayed as asterisks again.

Create Authentication Profiles

The WMI, Linux/Unix, and SNMP tabs allow you to create, edit, view, and delete authentication profiles for discovery. An authentication profile contains credential information necessary for discovery to access and gather information about computer systems and devices in your network.

You can create one or more authentication profiles under each of the WMI, Linux/Unix, and SNMP tabs.

Note: Creating authentication profiles is not required for discovery. However, only IP discovery is used if no authentication profiles exist, and information about discovered systems may be limited.
Select the WMI, Linux/Unix, or SNMP tab, and click the name of an authentication profile in the left pane to view its properties in the pane to the right.

To modify an existing authentication profile, select it and edit the fields as necessary, then click Save. To delete an authentication profile, click the trash can icon next to the name of the profile in the left pane, and click Save.

Follow these steps to create an authentication profile:

1. Click New credentials in the left pane.
2. Enter information in all of the required fields.
   Required fields are outlined in red.
3. Click Next.
   The information you enter is saved when you click Next and move through the Discovery Wizard.

**Linux/Unix**

Linux/Unix authentication profiles use SSH or Telnet to access and discover Linux and Unix systems.

**Description**

Name for the authentication profile.

**ID**

This read-only field is the Nimsoft system ID for this authentication profile, assigned when the profile is saved. It identifies the profile uniquely for re-use in other areas of USM that reference authentication profiles.

**User**

User name.

**Password**

The user password. Check the Show new passwords check box to verify the text as you enter it.

**SSH or Telnet**

Select the communication protocol to use, SSH (Secure Shell) or Telnet (no secure authentication or encryption).

**Note:** Discovery Agent uses password authentication to connect to a target device over SSH. Discovery Agent cannot communicate with a device where SSH is configured for other authentication methods, such as keyboard-interactive or challenge-response authentication.
SNMP

CA Nimsoft Discovery supports SNMP versions 1, 2c, and 3. SNMP v3 adds security features which v1 and v2c lack. As a result, authentication profile configuration fields in the Discovery Wizard that deal with security and privacy (encryption) are only active when 3 is selected in the Version pull-down menu.

We recommend the following best practices:

- Create a minimal set of SNMP authentication profiles that will, in aggregate, provide SNMP access to all your network devices and hosts that support SNMP.

- Set up as many of your network devices as possible to use "universal" read-only credentials. For example, you could define a read-only (get-only) credential to be nms_get_only. Then set up every device possible to allow read-only SNMP access via this universal credential. This minimizes the number of SNMP authentication credentials that must be attempted on network nodes, and simplifies your discovery configuration.

- If there are devices that accept unique SNMP credentials, create one authentication profile for each of those. You can specify a unique port within the range of 1 to 65535 for the profile. If no port is specified, the default port 161 is used.

<table>
<thead>
<tr>
<th>Field (SNMP v1 or v2)</th>
<th>Required</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Description</td>
<td>Yes</td>
<td>Name for the authentication profile</td>
</tr>
<tr>
<td>ID</td>
<td></td>
<td>This read-only field is the Nimsoft system ID for this authentication profile, assigned when the profile is saved. It identifies the profile uniquely for re-use in other areas of USM that reference authentication profiles.</td>
</tr>
<tr>
<td>Version</td>
<td>Yes</td>
<td>The SNMP version supported by the monitored device. When version 1 or 2 is selected, only the Community field is active.</td>
</tr>
<tr>
<td>Community</td>
<td>Yes</td>
<td>The SNMP community string. Check Show new passwords to verify the text as you enter it. Be aware that this string is sent across the network in clear text as part of SNMP v1 or v2c requests, which may pose a security risk.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Field (SNMP v3)</th>
<th>Required</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Description</td>
<td>Yes</td>
<td>Name for the authentication profile</td>
</tr>
<tr>
<td><strong>ID</strong></td>
<td><strong>This read-only field is the Nimsoft system ID for this authentication profile, assigned when the profile is saved. It identifies the profile uniquely for re-use in other areas of USM that reference authentication profiles.</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Version</strong></td>
<td><strong>Yes</strong></td>
<td><strong>SNMP version supported by the monitored device. Versions 1, 2c, and 3 are supported. When v3 is selected, other fields for security and privacy are enabled.</strong></td>
</tr>
</tbody>
</table>
| **Password** | **See note** | **The password associated with the SNMP v1/v2c device or SNMP v3 user. Check **Show new passwords** to verify the text as you enter it.**  
 **Note:** This field is enabled and required if either **AuthNoPriv** or **AuthPriv** security is selected. See the description for the Security field below. |
| **User** | **Yes** | **SNMP v3 user name used to access the monitored device. Required for all SNMP v3 security levels. See the description for the Security field below.** |
| **Method** | **Yes** | **SNMP v3 method of encryption, when **AuthNoPriv** or **AuthPriv** security is selected (see the description for the Security field below):**  
 ■ **None**  
 ■ **MD5** - MD5 Message-Digest Algorithm (HMAC-MD5-96)  
 ■ **SHA** - Secure Hash Algorithm (HMAC-SHA-96) |
| **Security** | **Yes** | **SNMP v3 security level of the user. Depending on what level of security is selected, other security fields are enabled or disabled:**  
 ■ **NoAuthNoPriv** - messages sent unauthenticated and unencrypted  
 ■ **AuthNoPriv** - messages sent authenticated but unencrypted  
 ■ **AuthPriv** - messages sent authenticated and encrypted |
| **Priv.Password** | **See note** | **SNMP v3 privacy password to use if **AuthPriv** security level is selected. Must be at least eight characters. Do not confuse with the user password (authentication).**  
 **Note:** This field is enabled and required if **AuthPriv** security is selected |
Priv. Protocol | See note | SNMP v3 privacy (encryption) protocol to use.
--- | --- | ---
- **DES** - Data Encryption Standard
- **AES** - Advanced Encryption Standard

**Note:** Enabled and required if **AuthPriv** is selected.

### WMI

WMI (Windows Management Interface) discovery scans servers and hosts running Windows to gather system information. WMI discovery runs only on discovery agents hosted on Windows systems.

**Description**

Name for the authentication profile.

**ID**

This read-only field is the Nimsoft system ID for this authentication profile, assigned when the profile is saved. It identifies the profile uniquely for re-use in other areas of USM that reference authentication profiles.

**User**

User name, in the form of `Domain\user name`. `user_name` and `IP_address\user_name` are also allowable.

**Password**

User password. Check the **Show new passwords** check box to view the text as you enter it.

### Define Ranges

Use the Ranges tab of the Discovery Wizard to define network addresses, ranges, or masks where devices are to be discovered. At least one network range must be entered for discovery to run.

You can assign any combination of SNMP, Linux/Unix, and WMI authentication profiles to a range. The discovery process records any device within a range that responds to a request on any protocol, including a simple ICMP ping. This means you can include end nodes (such as servers, network printers, network storage systems, or workstations) in a range, even if they don’t respond to requests via SNMP or other management protocols.

If no authentication profile is assigned to a range, basic discovery is performed using protocols that do not require authentication, but discovery may not be complete and information about discovered systems is limited.
Best Practices for Creating Ranges

For each discovery agent, review the assigned ranges to minimize predictable timeouts. To optimize performance and avoid duplicate entries, each discovery agent should discover an exclusive part of the network.

Tips to decrease discovery run time:

- The discovery agent tries each credential on each IP address and waits for a timeout (or success) with each attempt. Use a single credential in a range that has a high probability of immediate success on the nodes in that range to speed up discovery.

- When you apply an authentication profile to a range, make sure that most, if not all, devices defined by that range will accept the authentication profile.

- If you include devices that do not respond to requests on any management protocol, place them in a discovery range with no authentication profiles assigned to the range.

- If you use SNMP for a device that accepts only a unique SNMP community string, create a Single type range and specify the device’s IP address. Assign the corresponding authentication profile to the range.

- When using SNMP, to avoid unnecessary authentication traps/alerts, assign only one SNMP authentication credential per discovery range.

Create a Range

Follow these steps:

1. Click New range in the left pane of the Ranges tab.
2. Enter a name for the range.
3. In the Range definition section, specify the area(s) of your network where you want to perform discovery.

- **Mask** - Bitmask for a subnet using Classless Inter-Domain Routing (CIDR) notation with a base IPv4 address and a routing prefix. For example, 195.51.100.0/24. The value /24 refers to a Class C subnet of 256 addresses. Other values for reference: /30 (4 addresses) and /16 (65,536 addresses, or a Class B subnet).

  Note: When you enter a subnet mask, the number of IP addresses the mask represents is displayed (the number of effective hosts minus two). Only /16 subnets or smaller are supported.

- **Range** - Range of IPv4 addresses.

- **Single** - Single IPv4 or IPv6 address. You can use abbreviated IPv6 address forms, and IPv6 addresses that refer to IPv4 addresses. However, IPv6 anycast, multicast, and loopback addresses, and default routes are not supported.

You can also click the Add multiple IPs icon (🔗) above the Range definition section. Copy and paste the IP addresses into the Import IPs dialog, one entry per line. After you click OK, any errors are highlighted in red.

4. Click **New IP range or single IP address** to add another IP range, address, or mask if desired.

5. In the Credentials section, you can assign authentication profiles to the selected range. By default, all of the authentication profiles are selected.

  If you have a large number of authentication profiles in the list, you can enter the name of a profile to filter the list.

  To view only the profiles that are selected, click the **Hide unused credentials** check box.

6. When you have finished defining ranges, click **Next**.

**Schedule Discovery**

In the Schedule tab, you can schedule discovery to run in the future, and/or you can run discovery immediately. You can schedule either a single discovery run or recurring runs.

A scheduled discovery does not interrupt a discovery that is already running. If at the time a discovery run is scheduled another discovery run is in progress, the scheduled discovery is ignored.

If you select **Run discovery now** and discovery is in progress, the current discovery run is terminated and the new run is executed.
Follow these steps to start and/or schedule discovery:

1. Leave the **Run discovery now** check box selected unless you do not want to run discovery when you complete the Discovery Wizard.
2. To schedule discovery, select the **Schedule discovery** check box.
3. Enter information in the date and time fields.
   - The time field is in 24-hour format. The time is the local time of the user.
4. To schedule recurring discovery runs, select the **Recurring every** check box, and enter the number of hours for the recurrence interval.
5. Click **Finish** to complete the Discovery Wizard.

**Troubleshooting the Discovery Wizard**

This section contains troubleshooting information for the Discovery Wizard.

**I Cannot Save My Information**

**Symptom:**
I cannot save my authentication profile or range.

**Solution:**
All required fields must have valid entries in order to save an authentication profile or range. Required fields are outlined in red.

**I See Messages About Exclude Scopes (Ranges)**

**Symptom:**
When I launch the Discovery Wizard, I see the following error message:
Exclude scopes (ranges) are no longer supported. Would you like to remove them?

**Solution:**
This occurs because ranges of IP addresses to be excluded from discovery were previously configured for your discovery agent. The Discovery Wizard does not support exclude ranges and cannot proceed until the exclude ranges are deleted. You can either accept the prompt to delete your exclude ranges, or exit the Discovery Wizard and manually delete them from the NIS database. If you need to view your exclude ranges, you can access them through the Nimsoft Remote Admin application.
Run File-based Import

Using file-based import, CA Nimsoft administrators can import device and host information into CA Nimsoft Monitor without network scans or manual entry. Because it is not necessary to scan the IT environment, file-based import of devices causes fewer security alerts, and can be faster than automated discovery using the Discovery Wizard.

**Note:** If a system is discovered by an automated scan of the network and is also included in a file-based import, the file-based import takes precedence. If information about the system differs, the information in the XML file for file-based import is the information that is stored in the database.

**Follow these steps:**

1. Create an XML file containing information about computers or network devices.
   
   For details about the contents of the XML file, see the help topic [XML File Schema](#) (see page 422).

2. Expand the **Discovery** node in the tree view in the Unified Service Manager.

3. Hover over the **External** node in the tree and click the import icon (騙), or click the **External** node and choose **Discovery Import** from the **Actions** menu.

4. Navigate to the XML file in the file browser, then click **OK**.
   
   The device information is imported into the Nimsoft database. Processing by discovery_server starts, and can take several minutes or more to finish.

5. To view imported devices, click the **External** node.
   
   The devices are displayed in the table to the right.

**Alternative import method:**

The cm_data_import probe monitors a directory for valid XML files, and if it finds one, it automatically imports the information into the database. Here is how the process works:

1. Copy the XML file you prepared to `<Nimsoft install directory>\Probes\Service\cm_data_import\import` directory on the system that hosts the cm_data_import probe.

2. The cm_data_import probe scans this directory at regular intervals (the default is 60 seconds).

3. If the probe finds a valid import file, it imports the device information in the file into the Nimsoft database.

4. The probe moves the file to a timestamped subfolder in the `<Nimsoft install directory>\Probes\Service\cm_data_import\processed` directory, also on the probe host, and logs the results of the process.
XML File Schema

This section describes how to create an XML file for use with file-base discovery.

The XML file must include these required properties for each host or device:

- **PrimaryIPv4Address** - List the IPv4 address. Although the PrimaryIPv6Address tag exists, IPv6 addresses are not currently supported in discovery.
- **Origin** - Setting the origin correctly is important. See details on the Origin property in the table below.

Here is an example of XML that illustrates how to import one device with IP address 1.2.3.4 and origin "MyOrigin" in the database.

```xml
<?xml version="1.0" encoding="UTF-8" standalone="yes" ?>
<DevicesToImport xmlns="http://nimsoft.com/2012/11/cm-data-import"
xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance">
  <Device>
    <PrimaryIPV4Address>1.2.3.4</PrimaryIPV4Address>
    <Origin>myOrigin</Origin>
  </Device>
</DevicesToImport>
```

Additional optional properties can be included, as shown in the example below. You can also find this example file, named example1MaximalDevice.xml, in the `<Nimsoft install directory>\Probes\Service\cm_data_import\schema` directory, located on the system that hosts the cm_data_import probe—typically the primary hub.
The following table describes the XML properties. For properties that refer to open enumerations, navigate to <Nimsoft installation path>\Probes\Service\cm_data_import\schema and open either usm-openenums.xml or cm-data-import-openenums.xml to view the defined values for each enumeration instance. It is strongly recommended you use values defined by the open enumerations, though not strictly required.

To deploy a robot to an imported system using USM and ADE, some additional properties beyond IP address and origin are required. These are noted in the table below.
<table>
<thead>
<tr>
<th>Property</th>
<th>Required?</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Origin</td>
<td>Yes</td>
<td>QoS data from probes are tagged with an origin name to identify the origin of the data. The origin name defaults to the Nimsoft hub name but can be overridden at the hub or robot (controller) in order to separate data in a multi-tenancy environment. To ensure that QoS data from probes is correlated to this device, the origin name specified here should match the origin name you intend to use in your Nimsoft infrastructure of hubs and robots.</td>
</tr>
<tr>
<td>Label</td>
<td>No</td>
<td>A short description or caption.</td>
</tr>
<tr>
<td>Description</td>
<td>No</td>
<td>Text description of the device.</td>
</tr>
<tr>
<td>PrimaryDnsName</td>
<td>No</td>
<td>The entity's Domain System Name, which may be used for correlation.</td>
</tr>
<tr>
<td>PrimaryIPV4Address</td>
<td>Either IPv4 or IPv6 is required</td>
<td>An IPv4 address for the entity that may be used for correlation and identity.</td>
</tr>
<tr>
<td>PrimaryIPV6Address</td>
<td>Either IPv4 or IPv6 is required</td>
<td>An IPv6 address for the entity that may be used for correlation and identity. The address is expressed using the formal, complete IPv6 notation (8 groups of up to 4 hex digits, using only uppercase where applicable, separated by colons).</td>
</tr>
<tr>
<td>Property</td>
<td>Required?</td>
<td>Description</td>
</tr>
<tr>
<td>-------------------</td>
<td>-----------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>OtherIPAddresses</td>
<td>No</td>
<td>An entity may have multiple IP addresses. This element captures the values of those addresses for correlation and identity. The various values are comma-separated. Either IPv4 or IPv6 values may be specified, but the addresses should be formatted following the regex patterns defined by usm-core:IPV4AddressFormat or usm-core:IPV6AddressFormat.</td>
</tr>
<tr>
<td>PrimaryMacAddress</td>
<td>No</td>
<td>A MAC address for the entity that may be used for correlation and identity. The address is expressed as 6 groups of 2 hex digits (using only uppercase), separated by dashes.</td>
</tr>
<tr>
<td>Property</td>
<td>Required?</td>
<td>Description</td>
</tr>
<tr>
<td>-----------------------</td>
<td>-----------</td>
<td>--------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>OtherMacAddress</td>
<td>No</td>
<td>An entity may have multiple MAC addresses. This element captures the values of those addresses, whereas the PrimaryMacAddress element is designed to be used for correlation. The various values are comma-separated and are formatted following the regex pattern defined by usm-core:MacAddressFormat.</td>
</tr>
<tr>
<td>PrimaryOSType</td>
<td>Required by ADE for robot deployment</td>
<td>OS type, defined by the open enumeration OSTypeEnum. For Linux, the Linux distribution name is required by ADE (for example, Linux-RedHat).</td>
</tr>
<tr>
<td>PrimaryOSVersion</td>
<td>No</td>
<td>OS version details.</td>
</tr>
<tr>
<td>ProcessorType</td>
<td>Required by ADE for robot deployment</td>
<td>Processor environment/type (such as &quot;x86&quot;) as defined by the open enumeration ProcessorEnvironmentEnum.</td>
</tr>
<tr>
<td>Vendor</td>
<td>No</td>
<td>The hardware vendor/manufacturer's name, as defined by the open enumeration VendorEnum.</td>
</tr>
<tr>
<td>Model</td>
<td>No</td>
<td>The hardware model name/number.</td>
</tr>
<tr>
<td>PhysSerialNumber</td>
<td>No</td>
<td>An identifying string assigned by the hardware manufacturer and printed on a tag attached to the component. The data for this element should be input directly from the manufacturer’s tag on the component (which may be an RFID tag), or read from the entPhysicalSerialNum field of SNMP's Entity-MIB. Note that a virtual entity would NOT have a PhysSerialNumber.</td>
</tr>
<tr>
<td>PrimaryDeviceRole</td>
<td>No</td>
<td>The device role as defined by the open enumeration DeviceRoleEnum.</td>
</tr>
<tr>
<td>PrimarySoftwareRole</td>
<td>No</td>
<td>The software role as defined by the open enumeration SoftwareRoleEnum.</td>
</tr>
<tr>
<td>DBServerType</td>
<td>No</td>
<td>The type of database server of which this is an instance, defined by the open enumeration DBServerTypeEnum.</td>
</tr>
<tr>
<td>Property</td>
<td>Required?</td>
<td>Description</td>
</tr>
<tr>
<td>-------------------------</td>
<td>-----------</td>
<td>--------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>AppServerType</td>
<td>No</td>
<td>The type of application server, as defined by the open enumeration AppServerTypeEnum.</td>
</tr>
<tr>
<td>VirtualizationEnvironment</td>
<td>No</td>
<td>Value indicating the specific virtualization environment (hypervisor manager) of a hypervisor or virtual system. Values are defined in the open enumeration VirtualizationTypeEnum.</td>
</tr>
<tr>
<td>WmiAuthId</td>
<td>Either WmiAuthId or ShellAuthId is required by ADE for robot deployment</td>
<td>A Nimsoft defined authentication profile ID to use for WMI access. This is the ID field in the WMI authentication profile.</td>
</tr>
<tr>
<td>ShellAuthId</td>
<td>Either WmiAuthId or ShellAuthId is required by ADE for robot deployment</td>
<td>A Nimsoft defined authentication profile ID to use for SSH or telnet access. This is the ID field in the Shell authentication profile.</td>
</tr>
<tr>
<td>SnmpAuthId</td>
<td>No</td>
<td>A Nimsoft defined authentication profile ID to use for SNMP access. This is the ID field in the SNMP authentication profile.</td>
</tr>
<tr>
<td>MonitorFrom</td>
<td>No</td>
<td>If the device will be remotely monitored, this specifies the system to monitor this device from. The value can be specified as an IP address, simple host name, fully qualified domain name or Nimsoft address (/NimsoftDomain/HubName/RobotName). A Nimsoft robot should be installed on the system specified here. If the robot is not installed, this device will not be imported. The origin name used by the robot should match the origin specified for this device to ensure that QoS data from probes is correlated with this device.</td>
</tr>
</tbody>
</table>
View Discovered Systems

The **Discovery** node in the tree view of the Unified Service Manager allows you to view computers and devices that have been discovered on your network.

The Discovery section of the tree contains discovery agents, with network ranges under each discovery agent. The tree also has an Automatic and an External node.

Icons next to the tree nodes help identify the type of node and provide additional information:

- Top-level Discovery node or discovery agent.
- Network range.
- Automatic. Some probes automatically discover systems, and those systems are displayed under this node.
- External. Systems listed under this node were imported using file-based discovery.
- A discovery is scheduled. Hover over the icon to see the next scheduled time in the tool tip.
- Discovery in progress. The proportion of blue indicates the progress of discovery.
- No discovery scheduled.

Click a node in the tree to view associated systems and their properties in the table to the right. To view properties for all discovered systems, click the **Discovery** node.

A pie chart above the table displays information about discovered systems for the selected node. Choose a different criterion (**Device Type**, **Operating System**, etc.) from the pull-down menu to change the data displayed in the pie chart.

Click a slice in the pie chart or an item in the chart legend to filter for those systems. Only the systems represented in the slice are displayed in the table and reflected in the response links to the right. Click the slice or legend item again to clear the filter.

The response links to the right of the pie chart list systems according to how recently they responded to a request from the discovery agent. Click one of these links, such as **Recent (last day)**, to filter for those systems. Only those systems are displayed in the pie chart and in the table. Click the link again to clear the filter.

**Note:** Systems that do not respond are eventually purged from the database. By default, 30 days after the last response from a system, the system is deleted from the database.
A Quick Filter field below the response links allows you to filter for text in the Name, IP Address, Domain, OS Name, and Origin columns of the table.

Click a column header to sort the table by the column.

A key icon (❖) in the table indicates a discovery agent was able to authenticate with the system using one of the defined authentication profiles. Hover over the key icon to view the type and name of the authentication profile used.

You can export data for a discovery agent or network range. The data includes more columns than are displayed in the Inventory table. Data is exported to a .csv file, which is saved in a location you choose. To export data, click a discovery agent or network range in the tree, then select Export Group from the Actions menu.

Note: When you choose Export Group, all systems for the selected discovery agent, or selected network range, are exported, regardless of whether you filtered the display in the Inventory view.

More Information about Discovery

For an overview of automated discovery, including discussion on configuration of the probe_discovery queue, the discovery_server, discovery_agent, and cm_data_import probes, see the Discovery User Guide.

Automatic Robot Deployment

CA Nimsoft Monitor administrators can use USM to deploy robots automatically to a group of systems or to an individual system. You can:

- Deploy robots to all, or selected, members of a group
- Deploy a robot to a specific system
- Do a Basic or Advanced Search for systems to deploy robots to
- Import an XML file listing systems to deploy robots to

Once you select the systems and start a deployment job, robots are automatically installed on the selected systems.

In order to select in the USM UI systems to deploy robots to, the system must have known credentials. Credential information is entered in the Nimsoft Remote Admin portlet and associated with systems during discovery. Systems that do not have known credentials cannot be selected in the UI for automatic robot deployment. When importing a list of systems from an XML file, however, the credentials are specified in the XML file and the systems do not need to have known credentials through the discovery process.
You must have the **USM Automatic Robot Installation** ACL permission in order to deploy robots automatically. Account contact users cannot deploy robots automatically. Also, the automated_deployment_engine probe version 1.10 or later must be running in your environment.

Administrators can also use either the GUI or the command-line interface for the automated_deployment_engine probe to automatically deploy robots. For more information on these methods, see the *CA Nimsoft Monitor Server Installation Guide*.

For a list of platforms that support automatic robot deployment, see the [Compatibility Matrix](#).

### Deploy Robots Automatically

CA Nimsoft Monitor administrators can deploy robots automatically to a group of systems or to an individual system. You must have the **USM Automatic Robot Installation** ACL permission in order to deploy robots automatically. Also, the automated_deployment_engine probe version 1.10 or later must be running in your environment.

**Note:** If you do not have the **USM Automatic Robot Installation** ACL permission, or if you are an account contact user, the **Deploy Robots** tab is not displayed.

**Follow these steps:**

1. **Do one** of the following:
   - In the tree view, click the group or system you want to deploy robots to.
   - In a badge view, drill down to the group or system you want to deploy robots to.
   - After doing a Basic or Advanced Search for systems you want to deploy robots to, click on the **Search Results** node in the tree view (to deploy robots to all the systems found) or click on a system under the **Search Results** node (to deploy robots to that system only).

2. **Choose** **Deploy Robots** from the **Actions** menu.

   The Administration dialog opens with the **Deploy Robots** tab selected and the group members or system(s) listed in the table.
3. Select the systems you want to deploy robots to.

To select all systems that have credentials, click the check box in the column header. To find specific systems in the list, enter part or all of the system name in the Find field. This highlights the first occurrence of the name in the table.

**Note:** A system must have known credentials in order to select it to automatically deploy robots to. If the system does not have known credentials, you cannot select the check box. Credentials are associated with systems during discovery. Credentials are specified in authentication profiles, which are assigned to a scope (portion of the network) to be discovered. If you want to deploy robots automatically to a system that does not have known credentials you can do one of the following:

- Run the discovery wizard for a scope that includes the system, making sure to assign an authentication profile to the scope.
- Import the system in an XML file, specifying the credentials in the XML file.

For more information on both of these options, see the section of help on Discovery (see page 412).

4. Choose a **Hub** for the robots to report to.

5. Enter an **Origin** for the robots.

   If you do not enter an origin, the name of the hub is assigned as the origin.

6. (Optional) Edit the **Job Description** for this robot deployment.

7. Click **Deploy**.

   Deployment begins and the Robot Deployment Status dialog opens. The table on the right displays messages about the current deployment job. The status is updated every 10 seconds.

8. Click **Close** when the deployment is complete.

---

**Deploy Robots to Systems Listed in a File**

CA Nimsoft Monitor administrators can import a list of systems to deploy robots to automatically. To do so, first define the list of systems in an XML file, then import the list into the **Deploy Robots** tab.

**Define the XML File**

The XML file must list each computer system where a robot is to be installed, along with the required robot attributes. Optional robot attributes may also be specified (see the table below).

**Note:** Windows host names must specify the domain as well as the host name (domain\host name).
The following is an example of XML that lists the required attributes.

```
<hosts>
  <host>
    <profile>Linux</profile>
    <arch>64</arch>
    <hostname>server1</hostname>
    <username>root</username>
    <password>nimsoft</password>
    <domain>customer1</domain>
    <hubip>101.101.101.101</hubip>
    <hub>primaryhub</hub>
    <hubrobotname>robot1</hubrobotname>
    <hubport>48002</hubport>
  </host>
  <host>
    <profile>Windows</profile>
    <arch>64</arch>
    <hostname>customer1\server2</hostname>
    <username>root</username>
    <password>nimsoft</password>
    <domain>customer1</domain>
    <hubip>10.10.10.10</hubip>
    <hub>primaryhub</hub>
    <hubrobotname>robot1</hubrobotname>
    <hubport>48002</hubport>
  </host>
</hosts>
```

The following table describes the robot attributes.

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Required?</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>profile</td>
<td>Yes</td>
<td>Operating system (OS) of the host. Enter either Linux or Windows. This value is case-sensitive.</td>
</tr>
<tr>
<td>arch</td>
<td>Yes</td>
<td>Whether the host OS is 32-bit or 64-bit. Enter either 32 or 64.</td>
</tr>
<tr>
<td>hostname</td>
<td>Yes</td>
<td>Name or IP address of the computer system where the robot is to be installed. For Windows systems, you must include the domain name: domain\host.</td>
</tr>
<tr>
<td>username</td>
<td>Yes</td>
<td>User name for logging onto the system. The user must be an administrative user (Windows) or a root user (Linux) for the host where the robot is to be installed.</td>
</tr>
<tr>
<td>password</td>
<td>Yes</td>
<td>Password for logging onto the system.</td>
</tr>
<tr>
<td>domain</td>
<td>Yes</td>
<td>Domain the robot will belong to.</td>
</tr>
<tr>
<td>Attribute</td>
<td>Required?</td>
<td>Description</td>
</tr>
<tr>
<td>-----------------------</td>
<td>-----------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>hubip</td>
<td>Yes</td>
<td>IP address of the CA Nimsoft Monitor primary hub.</td>
</tr>
<tr>
<td>hub</td>
<td>Yes</td>
<td>Name of the CA Nimsoft Monitor primary hub. This is automatically updated if the hub name changes.</td>
</tr>
<tr>
<td>hubrobotname</td>
<td>Yes</td>
<td>Name of the robot on the CA Nimsoft Monitor primary hub. This is automatically updated by the controller if the hub robot name changes.</td>
</tr>
<tr>
<td>hubport</td>
<td>Yes</td>
<td>Port number of the hub. The default is 48002.</td>
</tr>
<tr>
<td>tempdir</td>
<td>No</td>
<td>Path on remote system where the Automatic Deployment Engine (ADE) places temporary resources for installation.</td>
</tr>
<tr>
<td>rsakeyfile</td>
<td>No</td>
<td>Local path to RSA public key file used to connect via SSH to certain remote targets.</td>
</tr>
<tr>
<td>first_probe_port</td>
<td>No</td>
<td>If specified, the controller gives probes port numbers to use. Otherwise random unused ports are taken.</td>
</tr>
<tr>
<td>secondary_domain</td>
<td>No</td>
<td>Domain of the secondary hub. Typically this is the same as the domain of the primary hub.</td>
</tr>
<tr>
<td>secondary_hub</td>
<td>No</td>
<td>Name of the secondary hub. This is automatically updated if the hub name changes.</td>
</tr>
<tr>
<td>secondary_hubrobotname</td>
<td>No</td>
<td>Name of the robot on the secondary hub. This is automatically updated by the controller if the hub robot name changes.</td>
</tr>
<tr>
<td>secondary_hubip</td>
<td>No</td>
<td>IP address of the secondary hub.</td>
</tr>
<tr>
<td>secondary_hubport</td>
<td>No</td>
<td>Hub port of the secondary hub. The default is 48002.</td>
</tr>
<tr>
<td>secondary_hub_dns_name</td>
<td>No</td>
<td>Fully qualified DNS name of the system the hub runs on. If this value is specified it overrides the secondary_hubip, which is then used only as a cached value if the DNS lookup fails.</td>
</tr>
<tr>
<td>secondary_robotip_alias</td>
<td>No</td>
<td>NAT address the robot should use when connected to the secondary hub. Valid values are:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- <strong>same</strong> = the NAT address for the primary hub. This is the default value.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- An IP address</td>
</tr>
<tr>
<td>Attribute</td>
<td>Required?</td>
<td>Description</td>
</tr>
<tr>
<td>-----------------</td>
<td>-----------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>robot_mode</td>
<td>No</td>
<td>Whether the robot initiates or accepts communication with its hub.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ normal - The robot can accept communication from, and initiate communication with, its hub.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ passive - The robot cannot initiate communication with its hub. All contact must be initiated by the hub.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ offline - The robot does not initiate or expect communications from its hub.</td>
</tr>
<tr>
<td>robotname</td>
<td>No</td>
<td>Name for the robot to be installed. The default is the host name.</td>
</tr>
<tr>
<td>robotip</td>
<td>No</td>
<td>Local IP address of the robot. The robot will send using this interface only. This value is propagated to its probes.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>By default no value is specified, in which case the robot automatically finds its IP address. In some cases where the machine has multiple interfaces it is unpredictable which interface the robot will use to communicate with its hub. Enter a value if you want to set which interface is used.</td>
</tr>
<tr>
<td>robotip_alias</td>
<td>No</td>
<td>In a NAT environment, the robot uses this IP address to identify itself to its hub.</td>
</tr>
<tr>
<td>controller_port</td>
<td>No</td>
<td>Specify a value to override the default controller port, which is 48000.</td>
</tr>
<tr>
<td>spooler_port</td>
<td>No</td>
<td>Specify a value to override the default spooler port, which is 48001.</td>
</tr>
<tr>
<td>hub_dns_name</td>
<td>No</td>
<td>Name of the DNS server for the hub.</td>
</tr>
<tr>
<td>hubdomain</td>
<td>No</td>
<td>Domain of the hub.</td>
</tr>
<tr>
<td>os_user1</td>
<td>No</td>
<td>User tag to be sent with messages generated from the robot. Used by the controller when sending internal messages directly to the hub and by the spooler when any message is spooled.</td>
</tr>
<tr>
<td>os_user2</td>
<td>No</td>
<td>User tag to be sent with messages generated from the robot. Used by the controller when sending internal messages directly to the hub and by the spooler when any message is spooled.</td>
</tr>
<tr>
<td>Attribute</td>
<td>Required?</td>
<td>Description</td>
</tr>
<tr>
<td>--------------------------</td>
<td>-----------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>set_qos_source</td>
<td>No</td>
<td>Whether the probe should use the robot name as the qos source instead of the host name. Valid values are yes and no. The default is no. Note: This does not work for older probes.</td>
</tr>
<tr>
<td>system_uptime_qos</td>
<td>No</td>
<td>Whether to send asynchronous QoS when the robot is up or down. Valid values are yes or no. The default is no.</td>
</tr>
<tr>
<td>autoremove</td>
<td>No</td>
<td>Whether the robot should unregister itself from the hub after it terminates. Valid values are yes and no. The default value is no.</td>
</tr>
<tr>
<td>default_priority_level</td>
<td>No</td>
<td>Default priority level for probes. Probes with lower priority levels are started first, with a delay between each priority level. This functionality is made redundant by the new start_after property of the probes.</td>
</tr>
<tr>
<td>proxy_mode</td>
<td>No</td>
<td>If set to 1, the controller acts as a proxy for all its probes so that all callback functions to the robot or its probes can be performed through the controller port. Valid values are 0 (off) and 1 (on). The default is 0.</td>
</tr>
<tr>
<td>proxy_log</td>
<td>No</td>
<td>Logging level for proxy functions in the controller.log file. Valid values are 0-8; the default value is 4. Typically values 1-5 are used. The controller.log file is located in the installation directory for the target robot.</td>
</tr>
<tr>
<td>hub_update_interval</td>
<td>No</td>
<td>Interval, in seconds, at which the controller should send alive or probelist information to the hub. The default is 900.</td>
</tr>
<tr>
<td>loglevel</td>
<td>No</td>
<td>Level of logging, 0-8, for messages from the robot. The default is 0.</td>
</tr>
<tr>
<td>logsize</td>
<td>No</td>
<td>Size, in kilobytes, for the controller.log file to reach before it is copied and truncated. The default is 100. The controller.log file is located in the installation directory for the target robot.</td>
</tr>
<tr>
<td>logfile</td>
<td>No</td>
<td>Name for the log file. The default is controller.log.</td>
</tr>
<tr>
<td>config_lock_timeout</td>
<td>No</td>
<td>Timeout, in seconds, for the configuration file lock. The default is 360.</td>
</tr>
<tr>
<td>port_alive_check</td>
<td>No</td>
<td>Interval, in seconds, for port alive checking. The default is 330.</td>
</tr>
<tr>
<td>Attribute</td>
<td>Required?</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------------------</td>
<td>-----------</td>
<td>--------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>port_alive_include_local</td>
<td>No</td>
<td>Whether to include ports registered from local probes when performing the port check. Valid values are yes or no. The default is yes.</td>
</tr>
<tr>
<td>startup_timeout</td>
<td>No</td>
<td>When there is a hub on the robot, this is the time, in seconds, the controller will wait for the local hub to start until other probes are started. If this timeout is reached, the robot does a failover. The default value is 900.</td>
</tr>
<tr>
<td>suspend_on_loopback_only</td>
<td>No</td>
<td>Suspend all probes if loopback is the only network. Valid values are yes or no. The default is yes.</td>
</tr>
<tr>
<td>temporary_hub_broadcast</td>
<td>No</td>
<td>If set to yes, only failover to the secondary hub is allowed; no broadcast occurs if the secondary is not available. Valid values are yes or no. The default is no.</td>
</tr>
<tr>
<td>do_not_broadcast</td>
<td>No</td>
<td>Same as temporary_hub_broadcast except broadcast is also disallowed for finding a primary hub. Valid values are yes or no. The default is no.</td>
</tr>
<tr>
<td>unmanaged_security</td>
<td>No</td>
<td>When broadcasting for a hub, this setting determines which hubs are candidates for establishing contact. Possible values are:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- <strong>not_locked</strong> - The controller can broadcast to any hub.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- <strong>none</strong> - The controller can broadcast to any hub.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- <strong>open</strong> - The controller can broadcast to any hub.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- <strong>domain_locked</strong> - The controller can broadcast only to hubs in its current domain.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- <strong>domain</strong> - The controller can broadcast only to hubs in its current domain.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- <strong>hub_locked</strong> - No broadcast is allowed. The controller can only contact its hub.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- <strong>hub</strong> - No broadcast is allowed. The controller can only contact its hub.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>The default is <strong>domain_locked</strong>.</td>
</tr>
<tr>
<td>Attribute</td>
<td>Required?</td>
<td>Description</td>
</tr>
<tr>
<td>------------------------</td>
<td>-----------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>send_alive</td>
<td>No</td>
<td>Whether or not to send alive messages to the hub. Possible values are:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ 1 - Send alive message to hub at each hubupdate interval.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ 0 - Do not send alive messages. Use this, for example, when running in an offline mode where you want contact between the robot and hub to be established only when needed.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ -1 - Use this if the alive message format is not understood by the (very old) hub. This sends the complete probe list instead.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>The default is 1.</td>
</tr>
<tr>
<td>alarm_level_comfail_restart</td>
<td>No</td>
<td>Whether to send an alarm, and at which severity level, when the controller is unable to communicate with a probe (that has a registered port). After the fourth attempt this alarm is sent and the probe is restarted. Possible values are:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ no alarm</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ clear</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ informational</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ warning</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ minor</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ major</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ critical</td>
</tr>
<tr>
<td></td>
<td></td>
<td>The default is no alarm.</td>
</tr>
<tr>
<td>alarm_level_dispatch_error</td>
<td>No</td>
<td>Severity level for an internal alarm indicating a socket system failure. This alarm is sent only if the loglevel setting is set to a number greater than 0. Possible values are:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ clear</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ informational</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ warning</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ minor</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ major</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ critical</td>
</tr>
<tr>
<td></td>
<td></td>
<td>The default is major.</td>
</tr>
<tr>
<td>Attribute</td>
<td>Required</td>
<td>Description</td>
</tr>
<tr>
<td>------------------------------</td>
<td>----------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
</tbody>
</table>
| alarm_level_max_restarts     | No       | Severity level for an alarm issued when a probe is restarted 10 times and quickly terminated each time. Possible values are:  
|                              |          | ■ clear  
|                              |          | ■ informational  
|                              |          | ■ warning  
|                              |          | ■ minor  
|                              |          | ■ major  
|                              |          | ■ critical  
|                              |          | The default is **major**.                                                    |
| alarm_level_start_error      | No       | Whether to send an alarm, and at which severity level, when unable to start a probe. Possible values are:  
|                              |          | ■ no alarm  
|                              |          | ■ informational  
|                              |          | ■ warning  
|                              |          | ■ minor  
|                              |          | ■ major  
|                              |          | ■ critical  
|                              |          | The default is **major**.                                                    |
| alarm_level_suspended        | No       | Whether to send an alarm, and at which severity level, when aborting the start of a probe because the robot state is **suspended**. Possible values are:  
|                              |          | ■ no alarm  
|                              |          | ■ clear  
|                              |          | ■ informational  
|                              |          | ■ warning  
|                              |          | ■ minor  
|                              |          | ■ major  
|                              |          | ■ critical  
<p>|                              |          | The default is <strong>no alarm</strong>.                                                 |</p>
<table>
<thead>
<tr>
<th>Attribute</th>
<th>Required?</th>
<th>Description</th>
</tr>
</thead>
</table>
| alarm_level_timed_not_finished | No        | Severity level of an alarm issued when a timed probe is not completed at the next scheduled start time. The timed probe is restarted when this occurs. Possible values are:  
  ■ clear  
  ■ informational  
  ■ warning  
  ■ minor  
  ■ major  
  ■ critical  
The default is **warning**.                                                                                       |
| alarm_level_timed_error_return | No        | Whether to send an alarm, and at which severity level, when a timed probe does not return 0 on termination. Possible values are:  
  ■ no alarm  
  ■ informational  
  ■ warning  
  ■ minor  
  ■ major  
  ■ critical  
The default is **warning**.                                                                                       |
| alarm_level_unregister        | No        | Severity level of alarm issued when a probe unregisters its port but does not terminate. Possible values are:  
  ■ clear  
  ■ informational  
  ■ warning  
  ■ minor  
  ■ major  
  ■ critical  
The default is **major**.                                                                                       |
<table>
<thead>
<tr>
<th>Attribute</th>
<th>Required?</th>
<th>Description</th>
</tr>
</thead>
</table>
| alarm_level_request_error | No        | Whether to send an alarm, and at which severity level, when unable to issue a request to distsrv. Possible values are:  
|                           |           | ■ no alarm  
|                           |           | ■ informational  
|                           |           | ■ warning  
|                           |           | ■ minor  
|                           |           | ■ major  
|                           |           | ■ critical  
|                           |           | The default is **major**.                                                                                                                   |
| alarm_level_postinstall   | No        | Whether to send an alarm, and at which severity level, when unable to start a post-install script on completing a package distribution and when the post-install script does not return the OK (0) status. Possible values are:  
|                           |           | ■ no alarm  
|                           |           | ■ clear  
|                           |           | ■ informational  
|                           |           | ■ warning  
|                           |           | ■ minor  
|                           |           | ■ major  
|                           |           | ■ critical  
|                           |           | The default is **no alarm**.                                                                                                                  |
| audit                     | No        | When and where to send audit messages. Possible values are:  
|                           |           | ■ post – Send audit message on controller events  
|                           |           | ■ yes – Log controller events to local file  
|                           |           | ■ post_detail – Send audit event on controller events and configuration file changes  
|                           |           | ■ file_detail – Log controller events and file change events to local file  
| audit_max_config_size     | No        | Maximum size, in bytes, of configuration file for content comparison.  
| audit_checkpoint_count    | No        | Number of versions of configuration files to retain.  

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Required?</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>alarm_timeout</td>
<td>No</td>
<td>Interval, in minutes, at which alarms for probes in the error state are repeated.</td>
</tr>
<tr>
<td>wait_after_unregister</td>
<td>No</td>
<td>Wait time, in seconds, after a probe unregisters a port. If the process is still running the unregister alarm is issued and the probe is set to the error state.</td>
</tr>
<tr>
<td>tz_offset</td>
<td>No</td>
<td>Timezone offset override in seconds, positive or negative.</td>
</tr>
<tr>
<td>config_locking</td>
<td>No</td>
<td>Signals the configuration tools that they should acquire a lock when starting relinquish it on termination. Possible values are yes or no. The default is no.</td>
</tr>
<tr>
<td>capture_output</td>
<td>No</td>
<td>Creates pipes for each started probe to capture any output they send to stdout or stderr. This output is appended to the probes log file. Enabling this functionality introduces an extra overhead in the probe. On Windows systems the probes must inherit resources from the controller and in some situations this may cause the controller to not terminate properly. Possible values are yes or no. The default is no.</td>
</tr>
<tr>
<td>default_fail_window</td>
<td>No</td>
<td>Number of seconds when a probe is started that it needs to run before the restart counter is reset for the probe. The default is 15. The default_fail_window set for the robot applies to all probes managed by the robot. If you want a different setting for a certain probe, you can set the fail_window setting in the configuration window for the probe.</td>
</tr>
<tr>
<td>max_restarts</td>
<td>No</td>
<td>Number of starts allowed before a probe is set to the error state. The restart counter is reset for a probe if it runs longer than its fail_window.</td>
</tr>
</tbody>
</table>
**Import the File**

CA Nimsoft Monitor administrators can import a list of systems to deploy agents (robots) to automatically. You must have the **USM Automatic Robot Installation** ACL permission in order to deploy agents automatically. Also, the automated_deployment_engine probe version 1.10 or later must be running in your environment.

**Note:** If you do not have the **USM Automatic Robot Installation** ACL permission, or if you are an account contact user, the **Deploy Agents** tab is not displayed.

**Follow these steps:**

1. Click the Administration icon ( ).
   
   The Administration dialog opens.

2. Click the **Deploy Agents** tab.

3. Click **Import**.

4. Browse to an appropriate XML file and click **Open**.
   
   The systems listed in the file are listed in the **Deploy Agents** table.

5. (Optional) Edit the **Job Description** for this agent deployment.

6. Click **Deploy**.
   
   Deployment begins and the Agent Deployment Status dialog opens. The table on the right displays messages about the current deployment job. The status is updated every 10 seconds.

7. Click **Close** when the deployment is complete.

**View the Status of Robot Deployment**

You can view status messages for automatic robot deployment jobs, including jobs that have completed.

**Follow these steps:**

1. Click the Administration icon ( )

2. Click the **Deploy Robots** tab.

3. Click **Deployment Status**.

   The Robot Deployment Status window opens.

4. In the table on the left, click the job you want to see messages for.

   Status messages for the job are displayed in the table on the right.
Getting Started with Service-Oriented Configuration

Service-oriented configuration allows you to define what you want to monitor in a monitoring template and then apply the template to a group of devices. The USM portlet automatically deploys the probes that support the monitors in the template and configures the appropriate probe settings. This eliminates the need to manually deploy and configure probes.

Not all probes support service-oriented configuration. For a list of probes that do, see SOC-Supported Probes.

Probes can also be configured through a configuration interface, accessed using Infrastructure Manager.

**Important:** Service-oriented configuration of the probes is not compatible with standard configuration through the probe's configuration interface. Do not mix or interchange configuration methods. If you do, the result will be unpredictable monitoring of your devices.

The above notice has these implications:

- If you use service-oriented configuration to configure a probe, thereafter you must not make any changes whatsoever to the configuration through the probe's native configuration interface!
- If you want to revert a probe from service-oriented configuration to standard configuration, contact Nimsoft support for assistance.

Set Up Service-Oriented Configuration

**Follow these high-level steps:**

1. Download the service-oriented configuration probes you want to use from the Nimsoft web archive to your local archive on your CA Nimsoft Monitor server.

   The probes must be in your local archive in order to see the monitors for the probes in the list of available monitors when creating monitoring templates in USM.

   **Note:** Service-oriented configuration can be used only with probes that support it. For a list of probes that support service-oriented configuration, see the SOC-Supported Probes topic in the SOC Probes Configuration Reference (SOC-probes/index.htm).

Getting Started with Service-Oriented Configuration

3. **USM portlet** – For existing implementations, set the device state to Managed for systems you want to monitor.
   
   This is not necessary for new implementations where discovery has not run (assuming you select the "unmanaged systems should be treated as managed" option during NM installation). For more information, see [Migration for Legacy Users](#) (see page 228).

4. **USM portlet** – Create groups.

5. **USM portlet** – Create monitoring templates.

6. **USM portlet** – Assign monitoring templates to groups.

**SOC Probes Configuration Reference**

This document describes configuration settings for probes configured using service-oriented configuration. To view this documentation, see the [SOC Probes Configuration Reference](http://docs.nimsoft.com/prodhelp/en_US/UMP/7.5/SOC-probes/index.htm).

Not all probes support service-oriented configuration, but more and more do. For a current list of probes that do, see the SOC-Supported Probes topic at the top of the document.

Probes can also be configured through a configuration interface, accessed using Infrastructure Manager. For configuration information using the native configuration interfaces, see the [Nimsoft Probe Library](http://docs.nimsoft.com/prodhelp/en_US/Probes/Catalog/index.htm).

**Important:** Service-oriented configuration of the probes is *not* compatible with standard configuration through the probe’s configuration interface. Do not mix or interchange configuration methods. If you do, the result will be unpredictable monitoring of your devices.

The above notice has these implications:

- If you use service-oriented configuration to configure a probe, thereafter you must *not* make any changes whatsoever to the configuration through the probe’s native configuration interface!

- If you want to revert a probe from service-oriented configuration to standard configuration, contact [Nimsoft support](#) for assistance.
Working with Groups

Groups are lists of computer systems. Administrators can create groups to assign the same monitoring template or report template to multiple computer systems.

There are three types of groups:

**Container**

A parent to other groups.

**Dynamic**

Contains the computer systems that meet a specified set of criteria. The membership of dynamic groups is updated automatically every 15 minutes. You can change the update interval in the nis_server probe configuration in Infrastructure Manager. Dynamic groups can be created manually, or UMP can automatically create groups based on OS Type, OS Name, Origin, User Tag 1 or User Tag 2.

**Static**

Contains a specified list of computer systems. Once you specify the list of systems that are members of the group, the membership does not change unless you manually add or remove systems.

You can use groups to organize your infrastructure by location, technology, or even a service. For example, you could create a container group named e-Commerce, a business service. The e-Commerce group could contain three groups named Web Servers, Application Servers, and Database Servers. These could be either dynamic or static groups and would contain servers to support each aspect of the e-Commerce service.

**Note:** A discovery agent must be running in your environment to discover the devices that can be added to groups.

**Note:** You must have the **USM Group Modification** permission set in your Access Control List (ACL) in order to create, edit, and delete groups.
Create a Group

You can create a group so that you can apply the same monitoring settings or report templates to all devices in the group.

Follow these steps:

1. In a badge or tree view, click on a container group and choose Add Group from the Actions menu. Or, in the tree view mouse over the name of a container group and click the Add Group icon (\.).

   **Note:** The Add Group option is available only for container groups.

   The Edit Group <Group Name> dialog is displayed.

2. In the Properties tab, edit the settings as needed:

   Type
   
   Choose Container, Dynamic, or Static.

   Name
   
   Enter a name for the group.

   Description
   
   Enter a description of the group.

   Priority
   
   Assign a priority. For more information, see Priority (see page 458).

   Monitoring Template State
   
   Choose whether or not monitoring templates assigned to the group are active. When templates are Active, all templates assigned to the group are applied to all members of the group. When templates are Inactive, none of the templates are applied to members of the group. However, the same monitoring templates remain active for other groups.

   Account
   
   Choose No Account if you do not want account contact users to see the group. Or, choose an account so that only account contact users assigned to that account or regular CA Nimsoft users can see the group (account contact users from other accounts cannot see the group).

   **Note:** You cannot set Priority or Monitoring Template State for container groups.
3. For dynamic or static groups, you can create a filter.
   For a static group, create a filter to help find the systems to be assigned to the group. Once the **Members** table is populated, select the systems to include in the group in the **Included** column.
   For a dynamic group, create a filter that defines the systems to be in the group.
   For information on filters, see **Filters** (see page 449).

4. Click **OK**.

5. Click the **Monitoring Templates** tab to assign monitoring templates to the group. For more information, see **Working with Monitors** (see page 456).

6. Click the **Report Templates** tab to assign report templates to the group. For more information, see **Reports** (see page 490).

**Automatic Groups**

Unified Service Manager (USM) can create certain dynamic groups automatically. You can create dynamic groups automatically according to:

**OS Type**

Creates groups according to type of OS, such as Windows, Unix, and so on.

**OS Name**

Creates groups according to name of operating system. For Unix systems, for example, creates groups such as Linux, Solaris, AIX, or HP-UX. For Windows systems, creates groups such as Windows 7, Windows XP, and so on.

**Origin**

Creates groups according to origin. The origin is a name assigned to QoS data from probes to identify the origin of the data. If you are an MSP, for example, typically the origin is the name of each customer. For enterprise customers, typically the hub name is used. The origin name can be set in the **Origin** field under **Advanced Settings** in the hub configuration interface in Infrastructure Manager. If no origin is assigned, the hub name is used.

**User Tag 1, User Tag 2**

Creates groups according to user tags. User tags are text fields that you define on the robot. For example, user tags could be used for geographic location or type of server.

If you create groups automatically by origin, for example, USM creates one dynamic group for each origin you have specified. As with other dynamic groups, membership is updated automatically, every 15 minutes by default.
It is recommended that you enter a name in the Parent Group field. This creates a container group for the automatic groups. If you leave the Parent Group field blank, the automatic groups are created under the root Groups node.

For example, if you are an MSP and are creating automatic groups by origin, you might name the parent group Customers.

You cannot create multiple levels of child groups for automatic groups; only one level of parent-child relationship is supported. If you enter the name of an existing group in the Parent Group field, the automatic groups are added to that parent group.

**Note:** When entering the name of an existing group in the Parent Group field, it is recommended that the group be a container group.

UMP comes with automatic groups created by OS Type under the parent group Servers.

Choosing more than one type of automatic group does not combine the types; each property you select creates groups for that property. For example, if you retain the OS Type with Parent Group Servers and also select Origin, with Parent Group Customers, groups for OS Type are created under the Servers group and groups for Origin are created under the Customers group.

You can view settings for automatic groups by clicking on an automatic group in the tree or badge view and choosing Edit Group from the Actions menu. However, settings for automatic groups cannot be modified.

**Note:** You cannot assign automatic groups to accounts. The Priority for automatic groups is 0 (highest priority) and the Monitoring Template State is Active.

---

**Create Groups Automatically**

USM can automatically create dynamic groups according to certain criteria.

In order to create groups automatically, you must not be an account contact user. For account contact users, the Configure Automatic Groups tab is not active.

**Follow these steps:**

1. Click the Administration icon (hammer) in Unified Service Manager. The Administration dialog opens.
2. Click the Configure Automatic Groups tab.
3. Select the type of automatic groups to create.
   **Note:** Choosing more than one type of automatic group does not combine the types; each property you select creates groups for that property.
4. If you want to create a group to be a parent of the automatic groups, enter a name in the Parent Group field.

For example, if you are an MSP and are creating automatic groups by origin, you could name the parent group Customers. If you leave the Parent Group field blank, the automatic groups are created under the root node.

5. Click OK.

Filters

For static groups, create the filter and then choose which of the listed systems to include in the group. For dynamic groups, create a filter to specify which systems are in the group. The list of members in the dynamic group is automatically updated every 15 minutes.

Note: If you filter on origin, the filter will only match systems using the discovered origin, the first origin listed in the device details. The filter will not apply to any enriched origins listed below the discovered origin.

Create a Filter

Create a filter to define the members of a dynamic or static group. For a dynamic group, the filter defines which devices are in the group. For a static group, the filter helps you find which devices to include in the group.

Note: The special characters \, %, and _ may need to be preceded by or enclosed in escape characters when you use basic search or advanced search, or when you apply filters in USM or in PRD. In the case of advanced search and filters, this applies when the contains, starts with, or ends with operator is selected. The escape character required depends on the type of database. For MySQL, the backslash (\) is the escape character; for MS SQL, special characters are enclosed in square brackets ([{}]). Searching for special characters is not supported for Oracle.

The following table shows how to enter special characters for each database type.

<table>
<thead>
<tr>
<th>Character</th>
<th>MySQL</th>
<th>Oracle</th>
<th>MS SQL</th>
</tr>
</thead>
<tbody>
<tr>
<td>\</td>
<td>\</td>
<td>Not applicable</td>
<td>\ (No escape character required)</td>
</tr>
<tr>
<td>%</td>
<td>%</td>
<td>Not applicable</td>
<td>[%]</td>
</tr>
<tr>
<td>_ (underscore)</td>
<td>_</td>
<td>Not applicable</td>
<td>[]</td>
</tr>
</tbody>
</table>
Follow these steps:

1. Do one of the following:
   - In a badge view, display a group and choose **Edit Group** from the **Actions** menu.
   - In tree view, click on a group and choose **Edit Group** from the **Actions** menu, or click the Edit Group icon (antasy) next to the group name in the tree.
   
   The Edit Group dialog is displayed.

2. In the **Filters** section of the **Properties** tab, choose items from the pull-down menus and enter text in the text field as appropriate.

3. Click the **Add Filter** icon (antasy) to add additional rows to the filter if wanted.

4. When you have defined the rows for the filter, click the **Apply Filters** icon (antasy) to confirm the results in the **Members** table.

   Systems that have not yet been added to the group are shaded in gray. Not all systems that match the filter criteria may be displayed; up to 100 systems that match the filter are displayed.

   For dynamic groups, all systems that match the filter are included in the group, even if they are not displayed in the **Members** table when you click **Apply Filters**. To view all members in the group, save the group and click on the group in a badge or tree view. All members are displayed. If there are more than 100 members click the arrow icon (antasy) to page through the list.

   For static groups, existing group members are displayed in addition to the filter results. For example, you select 50 systems as members of a static group and save the group. Then you edit the group, creating a filter to add members to the group. Up to 100 systems that match the filter plus the 50 existing members are displayed in the **Members** table when you click **Apply Filters**. If more than 100 systems match the filter and you do not see those you want to add to the group, you may need to refine the filter criteria so the systems you want to add are displayed in the **Members** table.

5. If the filter is for a static group, click the check boxes in the **Included** column to choose the systems you want to add to the group.

6. Click **OK**.

   The filter is saved and the systems added to the group are no longer shaded in gray in the **Members** table.

**Example**

We have a client, called Client A, who has both Windows and UNIX systems. Client A wants a group containing only Windows systems that are web servers or SQL servers.
First we narrow down the list using a unique attribute. Client A's system names all start with `clienta` so we select the **Name** field, the **contains** operator, and enter `clienta` in the text field. Then we click the **Apply Filters** icon to see the results. Now we see only Client A's systems.

Next we narrow down the group to only Windows systems. We click the **Add Filter** icon to add a row of filter options. We select the **and** operator, the **OS Type** field, and the **is** operator. We enter `windows` in the text field, then click the **Apply Filters** icon. Now we see only Client A's Windows systems.

Next we narrow down the group to include only web servers and SQL servers. We click the **Add Filter** icon to add a row of filter options. We select the **and** operator, the **Name** field, and the **ends with** operator and enter `websrvr` in the text field.

We click the **Add Filter** icon to add another row of filter options. We select the **or** operator, the **Name** field, and the **ends with** operator, and enter `sqlsrvr` in the text field.
We click the **Apply Filters** icon and the list now contains only Client A’s Windows systems that are web servers or SQL servers.

---

### User Tags

You can filter on two user-defined criteria, called User Tags, in portlets that have filters. You can also automatically create groups according to User Tag.

**User Tag 1** and **User Tag 2** are text fields that are defined on the robot. You can enter any text in these fields. (Use Infrastructure Manager to open the configuration window for the robot.) Once you have done so, you can filter for these values in the Unified Service Manager, Performance Reports Designer, and List Designer portlets.

For example, you could use **User Tag 1** for geographic location and **User Tag 2** for type of server. For some systems you enter **Europe** for **User Tag 1** in the robot configuration, and for some systems you enter **Asia**. For **User Tag 2**, you enter either **production server** or **web server**.
You can then create reports for your regional systems by filtering on User Tag 1 in the Performance Reports Designer (in the Filters tab) or List Designer (in the filter fields of the New Group pane). To do so, you would set the filter to User Tag 1 is Europe or User Tag 1 is Asia.

In the Unified Service Manager portlet, you can create separate groups for your production and web servers and apply different monitoring templates to them. To create the groups, in the Filters section of the Edit Group dialog you would set the filter to User Tag 2 is production server for one group, and User Tag 2 is web server for the other group.

User Tag 1 and User Tag 2 can be used to filter for any system that has a CA Nimsoft Monitor robot running on it. You cannot use User Tag 1 and User Tag 2 to filter for network devices that do not have a CA Nimsoft Monitor robot.

For information about automatically creating groups according to User Tag, see Automatic Groups (see page 447).

Filter Menus and Buttons

Filters have these menus and buttons:

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>and/or pull-down menu</td>
<td>Choose the and or or operator to apply to this row of the filter definition. This operator is present only in the second and subsequent rows.</td>
</tr>
<tr>
<td>(Blank)/not pull-down menu</td>
<td>Choose not in order to search for all systems except those that meet this row of the filter definition. Otherwise, leave blank.</td>
</tr>
<tr>
<td>Criterion pull-down menu</td>
<td>Choose the criterion to filter for, such as Name, IP Address, and so on.</td>
</tr>
</tbody>
</table>
Operator pull-down menu | Choose the appropriate operator, such as is, contains, starts with, and so on.
--- | ---
Text field | Enter the appropriate text for the criterion you chose.
Add Filter/Remove Filter icons | Click to add or remove rows for the filter definition.
Move up/Move down icons | Click to move the row up or down. Filter rows are applied in sequential order.
Apply Filters icon | Click to see the results of applying the filter in the Members table. Although you see the list of systems in the Members table, the group is not saved until you click OK.

**Note:** The special characters \, %, and _ may need to be preceded by or enclosed in escape characters when you use basic search or advanced search, or when you apply filters in USM or in PRD. In the case of advanced search and filters, this applies when the contains, starts with, or ends with operator is selected. The escape character required depends on the type of database. For MySQL, the backslash (\) is the escape character; for MS SQL, special characters are enclosed in square brackets ([]). Searching for special characters is not supported for Oracle.

The following table shows how to enter special characters for each database type.

<table>
<thead>
<tr>
<th>Character</th>
<th>MySQL</th>
<th>Oracle</th>
<th>MS SQL</th>
</tr>
</thead>
<tbody>
<tr>
<td>\</td>
<td>\</td>
<td>Not applicable</td>
<td>\ (No escape character required)</td>
</tr>
<tr>
<td>%</td>
<td>%</td>
<td>Not applicable</td>
<td>[%]</td>
</tr>
<tr>
<td>_ (underscore)</td>
<td>\</td>
<td>Not applicable</td>
<td>[_]</td>
</tr>
</tbody>
</table>
View Group Assignments

You can use the **Groups** tab of Unified Service Manager to view which groups a computer system is assigned to. This can help you determine whether any monitor collisions occur and how to set priority for your groups.

**Follow these steps:**
1. In a badge or tree view, click on a computer system.
2. Click the **Groups** tab in the right pane.

The table displays the name of the group, the type of group (container, dynamic, or static), the priority set for the group, and the state set for the group (active or inactive).

Servers Group

The **Servers** group is an automatic group that comes preconfigured with UMP. The **Servers** group creates sub-groups according to OS Type (Windows, Unix, and so on).

You cannot edit the properties of the **Servers** group or its sub-groups. However, you can assign monitoring templates and report templates. For the sub-groups, you can also click the Apply Filters icon to view what the results will be the next time the group is updated.

Delete a Group

Delete a group if you no longer need it. For container groups, all sub-groups must be deleted before you can delete the container group. For automatic groups, you can delete the Parent (container) group if all the sub-groups are deleted or if you change the name of the **Parent Group** in the Configure Automatic Groups tab.

**Follow these steps:**
1. In a badge or tree view, click on a group that does not have child groups.
2. Choose **Delete Group** from the **Actions** menu. Or, in the tree view mouse over the name of the group and click the Delete Group icon.
Working with Monitors

Monitors are checkpoints that can be configured to monitor specific resources of a system (such as CPU usage, memory usage, or bandwidth utilization) or events that occur within a system (such as disk crash or network unavailability). You can choose which monitors to include in a monitoring template and you can set the properties for each monitor to meet your specific monitoring requirements.

Once you create monitoring templates, you can assign them to groups of systems. Unified Service Manager automatically deploys the probes that support the monitors in the template and configures the appropriate probe settings.

This section of the help tells you how to create and configure monitors, assign monitors to monitoring templates, and assign monitoring templates to groups. It also tells you how to override monitor settings, apply a monitor an individual computer system, and check monitor status.

Note: You must have the USM Edit Monitoring Templates permission set in the Access Control List (ACL) to create, edit, and delete monitors and monitoring templates. You must have the USM Modify Individual Monitors for Computer Systems permission in order to apply a monitor directly to a system (without a monitoring template).

Local vs Remote Monitors

Each monitor is either a local or remote monitor, indicated in parentheses after the name of the monitor. For example, you can choose either the CPU Usage (local) monitor or the CPU (remote) monitor. The terms local or remote mean:

- Local - The monitor runs on the target system (the system being monitored). A CA Nimsoft robot must be installed on the system.
- Remote - The monitor runs on a system other than the target. To use remote monitors the CA Nimsoft discovery agent must be running in your environment to discover the systems to monitor. For example, you may run a discovery agent on a hub. A CA Nimsoft robot must also be installed on the system. CA Nimsoft calls the system where the discovery agent and robot are running a monitoring station. Devices discovered by a monitoring station will be remotely monitored by the same monitoring station. (Note: You can use the UMP Remote Admin application to configure remote discovery agents.)

When you choose a remote monitor, the appropriate probe is automatically deployed to the corresponding Monitoring Station. In general, choose a local monitor if a robot is installed on the target system. Otherwise, choose a remote monitor.

Note: Do not run both the local and remote versions of a monitor on the same system. Note also that auto-deployment does not occur in the case of the VMware probe.
If you choose a local monitor but no robot is installed on the system, you will see a "no robot found" error in the Monitor Status (see page 469) table.

Understanding Precedence, Priority, and State

Precedence, Priority, and State determine which version of a monitor is used if the same monitor is assigned to a system multiple times.

Precedence rules determine which monitors are applied within hierarchically related (parent-child) groups.

The Priority setting is used to determine which monitors are used among groups that are not related.

The State settings determine whether any monitors are applied to the system.

The following sections explain each of these in more detail.

Precedence

Precedence rules determine which templates, and therefore monitors, are applied first in hierarchical groups. Hierarchical groups are those with parent-child relationships.

For example, we have the hierarchical groups in the following diagram.

```
▼ 📝 Customers (370)
 ▼ 🏌️ Pro Golf (4)
   ▼ 📉 DB Servers (4)
   ☑️ Mail (0)
```

The groups Customers, Pro Golf, and DB Servers are hierarchically related, as are Customers, Pro Golf, and Mail. The groups DB Servers and Mail are peer groups and are not hierarchically related.

In hierarchical groups, monitoring templates are inherited from ancestor groups. The templates assigned to the group are applied first, then templates from the parent group, then templates from the grandparent group, and so on. If multiple templates contain the same monitor, which may be configured with different settings, the monitor from the first template applied is the one that takes precedence.
In our example, the DB Servers group has its own monitoring templates, as well as those assigned to the Customers and Pro Golf groups. Monitoring templates assigned to the DB Servers group are assigned first, then templates from the Pro Golf group, and then templates from the Customers group. If, for example, monitoring templates assigned to DB Servers and Customers contain the same monitor, the monitor from the template assigned to the DB Servers group is used.

**Priority**

A group can have multiple monitoring templates assigned to it, and a system can belong to multiple groups. This means the same monitor, with different values defined, may be assigned to a system multiple times. This is called a monitor collision.

Within hierarchically related (parent-child) groups, monitor collisions are resolved by precedence rules. Among non-related groups, the **Priority** setting determines which version of a monitor is used. The monitor instance from the group with the highest priority is the instance that is used.

You can set the priority level for each group. The highest priority is 0, and the lowest priority is 10. When a group is created, by default it is assigned a priority of 0.

If monitoring templates containing the same monitor are assigned from two groups with the same priority, one instance of the monitor is selected at random. When this occurs, a message is written to the log file for the Automatic Configuration Engine (ACE). This log file can be viewed using the Log Viewer in Infrastructure Manager. Right-click on the ace probe and select **View Log**.

**Note:** Because it is not possible to predict which monitor will be used when monitors from groups of equal priority collide, it is recommended for new UMP installations that you monitor collision notices in the ACE log file and adjust the **Priority** settings to resolve any collisions that are of concern.

Monitoring templates are inherited from ancestor groups, but the priority setting is not. So, if the priority of the parent group is 0 and the priority of the child group is 2, the templates assigned to the parent group are inherited by the child group but the priority remains 2.

To view a list of groups assigned to a computer system and the priority for each group, see [View Group Assignments](#) (see page 455). To view the status of monitors assigned to a computer system, including whether there are any collisions, see [View Monitor Status](#) (see page 469).
Example

We have a computer system named Development that belongs to two groups, Group1 and Group2. Both groups have monitoring templates assigned to them that contain the Net_Connect monitor, but with different values for the configuration settings. Because Development belongs to both groups, the Net_Connect monitor is assigned to Development twice, so a collision occurs. However, Group1 has a priority of 3, and Group2 has a priority of 2. Therefore, the Net_Connect monitor associated with Group2 is used. If, however, Group1 and Group2 have the same priority assigned, one of the Net_Connect monitors is chosen at random.

Follow these steps:

1. Click on a group in a badge or tree view.
2. Choose Edit Group from the Actions menu. Or, in the tree view mouse over the name of the group and click the Edit Group icon (▲). The Edit Group dialog is displayed.
3. Click the Properties tab.
4. Set a value for the Priority field.
   - 0 is the highest priority, and 10 is the lowest.
5. Click OK.

State

You can use state settings to determine whether or not CA Nimsoft Monitor monitors a system. If you have a system or group of systems that are not ready for production or are scheduled for maintenance, for example, you can choose settings so that the monitors assigned to those systems are not activated.

There are two ways you can do this. You can set the State property for an individual computer system, or you can set the Monitoring Template State property for a group.

The State property determines whether or not CA Nimsoft Monitor manages the system. When CA Nimsoft Monitor manages a system, it automatically deploys and configures the monitors that have been assigned to that system.

The Monitoring Template State property determines whether or not the monitoring templates assigned to a group are active.
Set the State for a System

You can set the **State** property for a computer system to determine whether that system is **Managed**, **Unmanaged**, or in **Maintenance** status.

The **State** you assign to a system carries across all groups the system is a member of. For example, if you have a system that is a member of two groups and you set the **State** for the system to **Unmanaged**, none of the monitors assigned by the two groups are deployed to the system.

**Note:** This is the same **State** property that in earlier releases (before UMP 6.0) was set for each system in Nimsoft Remote Admin. You can still set the **State** property in Remote Admin; however you can now set the **State** property from within UMP.

Follow these steps:

1. Do **one** of the following:
   - In a badge view, click on a device badge.
   - In tree view, click on the device name in the tree.
   The System details are displayed.
2. Choose the state you want to assign to the device from the **State** pull-down menu.

**Managed**

CA Nimsoft Monitor deploys and configures monitors assigned to the system. The monitors collect data, and any alarms defined for the system are generated.

**Unmanaged**

CA Nimsoft Monitor does not deploy or configure monitors for the system, and removes any monitors that have been deployed.

**Maintenance**

Any monitors deployed to the system are temporarily inactive. The monitors remain on the system, but data is not collected and alarms are not generated. You can set the **State** to **Managed** to resume these functions.

Set the Monitoring Template State for a Group

You can use the **Monitoring Template State** property to choose whether monitoring templates are applied to a group.

When templates are **Active**, all templates assigned to the group are applied to all members of the group (unless a system has a **State** of **Unmanaged** or **Maintenance**). When templates are **Inactive**, none of the templates are applied to members of the group. However, the same monitoring templates remain active for other groups.
For example, you have a monitoring template named Basic assigned to groups named Web Servers, Database Servers, and Production Servers. If you set the Monitoring Template State property to Inactive for the Database Servers group, no templates are applied to the Database Servers group. The Basic template continues to be applied to the Web Servers and Production Servers groups.

Follow these steps:

1. Do one of the following:
   - In a badge view, display a group and choose Edit Group from the Actions menu.
   - In tree view, click on a group and choose Edit Group from the Actions menu, or click the Edit Group icon next to the group name in the tree.
   The Edit Group dialog is displayed.
2. Choose the Monitoring Template State from the pull-down menu.
   - Active
     Monitoring templates assigned to the group are implemented for systems that are members of the group (unless the State property for the system has previously been set to Unmanaged or Maintenance).
   - Inactive
     The monitoring templates assigned to the group are not implemented.
3. Click OK.

Applying Monitors to Groups

You can use monitoring templates to assign a set of monitors to a group of computer systems. This allows you to configure the monitor settings once and reuse the set of monitors across your environment.

Multiple monitoring templates can have the same monitor, and the same monitor can be used within a template multiple times with different configurations.

Note: Some probes, such as the VMware probe, allow only a single instance of a monitor per template. For monitors with this restriction, once the monitor is added to a template, if you click the Add Template icon again, that monitor definition is excluded from the list of potential monitors from which to choose.
Create a Monitoring Template

Create a monitoring template to configure a set of monitors that can then be applied to a group of systems.

Follow these steps:

1. Click the Administration icon (📝).
   The Administration dialog is displayed.
   
   **Note:** You must have the USM Edit Monitoring Templates ACL permission set in order to edit monitoring templates. If the permission is not set, the Administration pane is not visible. ACL permissions are set in the Account Admin portlet or Infrastructure Manager.

2. Click the Edit Monitoring Templates tab.

3. Click the Add Template icon (➕).
   A new template is added to the Template list with the name New Template.

4. Enter a name and description in the text fields.

5. Choose an account from the Account pull-down menu if you want to assign the template to an account.

6. Click the Save Template icon (💾).
   
   **Note:** If you make changes to the Name, Description, or Account fields, save the template. Other changes, such as adding or editing a monitor, are applied immediately and you do not need to save the template.

7. To add a monitor to the template, with the name of the template selected in the Template list, click Add.
   The Add Monitor dialog is displayed. The dialog lists the monitors available to you. This list is populated with the monitors for the probes listed in the hub’s Nimsoft Monitor Archive.

8. Choose a category of monitors from the pull-down menu to narrow the list of monitors displayed, if wanted. Or, click on a column header to sort by that column.
   For more information about monitors, such as an explanation of local vs. remote monitors, see Monitors (see page 456).

9. Click a monitor to select it, then click OK.
   A configuration dialog displays properties similar to those you see in the Infrastructure Manager probe configuration windows. The properties vary for each monitor. For information on the properties for a specific monitor, see the SOC Probes Configuration Reference (SOC-probes/index.htm).

   **Note:** The properties you set for a monitor will be in effect for all computer systems in the group(s) the monitoring template is assigned to, unless you set monitor overrides. For more information, see Set a Monitor Override (see page 468).
10. Enter values for the monitor properties, then click **OK**.
    The monitor is added to the list of monitors for the template.

11. Repeat these steps to assign additional monitors to the monitoring template if wanted.
    **Tip:** Select the name of a monitor, then click **Summary** to see a pop-up window listing some of the properties for the monitor.

12. When you are finished adding and configuring monitors, click **Close**.
    The Administration dialog closes.

**Assign a Monitoring Template**

Assign a monitoring template to a group to apply those monitors to the systems in the group.

**Follow these steps:**

1. In a badge or tree view, click on a group and choose **Edit Group** from the **Actions** menu. Or, in the tree view mouse over the name of a group and click the Edit Group icon ( ).
    The Edit Group <Group Name> dialog is displayed.

2. Click the **Monitoring Templates** tab.

3. In the **Available** list, click the name of the monitoring template you want to assign to the group.

4. Click the **<<** button to move the monitoring template to the **Linked** list.
    The monitoring templates in the **Linked** list are linked, or assigned to, the selected group.

5. Use the **Move Up** and **Move Down** buttons below the **Linked** list to rearrange the order of monitoring templates. If there is a collision, the monitor associated with the monitoring template higher in this list takes precedence.

**Note:** Applying monitors results in a restart of the probe. If you are applying monitors to a large group of systems, you may want to test the configuration on a small group of systems first to avoid multiple restarts for the large group. The exception to this is the VMware probe, which monitors its configuration file and restarts itself when you apply a monitor using Service Oriented Configuration.
Copy a Monitoring Template

You can copy an existing monitoring template. You may wish to do this if you want to create a monitoring template that is similar to an existing template.

Follow these steps:

1. Click the Administration icon ( ).
   The Administration dialog is displayed.

   **Note:** You must have the USM Edit Monitoring Templates ACL permission set in order to edit monitoring templates. If the permission is not set, the Administration pane is not visible. ACL permissions are set in the Account Admin portlet or Infrastructure Manager.

2. Click the Edit Monitoring Templates tab.

3. Click the name of the monitoring template you want to copy in the Template list.

4. Click the Copy Template icon ( ).
   A new template is added to the Template list with the name Copy of `<Template Name>`. The settings for the template are the same as the template you copied.

5. Edit the name and description as desired.

6. Choose an account from the Account pull-down menu if you want to assign the template to an account.

7. Click the Save Template icon ( ).

   **Note:** If you make changes to the Name, Description, or Account fields, save the template. Other changes, such as adding or editing a monitor, are applied immediately and you do not need to save the template.

8. To edit settings for a monitor already assigned to the template, click the name of the monitor, then click Edit.
   A configuration dialog displays properties similar to those you see in the Infrastructure Manager probe configuration windows. The properties vary for each monitor. For information on the properties for a specific monitor, see the SOC Probes Configuration Reference (SOC-probes/index.htm).

   **Note:** The properties you set for a monitor will be in effect for all computer systems in the group(s) the monitoring template is assigned to, unless you set monitor overrides. For more information, see Set a Monitor Override (see page 468).

9. Edit values for the monitoring properties as desired, then click OK.

10. To add a monitor to the template, click Add in the Edit Monitoring Templates tab.
    The Add Monitor dialog is displayed. The dialog lists the monitors available to you. This list is populated with the monitors for the probes listed in the hub’s Nimsoft Monitor Archive.
11. Choose a category of monitors from the **Category** pull-down menu to narrow the list of monitors displayed, if wanted. Or, click on a column header to sort by that column.

   For more information about monitors, such as an explanation of local vs. remote monitors, see Monitors (see page 456).

12. Click a monitor to select it, then click **OK**.

   A configuration dialog displays properties similar to those you see in the Infrastructure Manager probe configuration windows. The properties vary for each monitor. For information on the properties for a specific monitor, see the SOC Probes Configuration Reference (SOC-probes/index.htm).

   **Note:** The properties you set for a monitor will be in effect for all computer systems in the group(s) the monitoring template is assigned to, unless you set monitor overrides. For more information, see Set a Monitor Override (see page 468).

13. Enter values for the monitor properties, then click **OK**.

   The monitor is added to the list of monitors for the template.

14. Repeat these steps to edit or assign additional monitors to the monitoring template if wanted.

   **Tip:** Select the name of a monitor, then click **Summary** to see a pop-up window listing some of the properties for the monitor.

15. When you are finished editing the monitors, click **Close**.

   The Administration dialog closes.

---

### Reapply a Monitoring Template

You can reapply monitoring templates to an individual computer system. You may wish to do this if you have changed the hardware configuration for that computer system, or if you have edited a monitoring template and do not want to wait for the updated settings to be pushed out to that computer system.

**Follow these steps:**

1. **Do one** of the following:
   - In the tree view, click the system you want to reapply a monitoring template to.
   - In a badge view, drill down to the system you want to reapply a monitoring template to.

2. Click the **Monitoring** tab.

3. Right-click in the table.
4. Choose **Update Monitoring Now** in the pop-up menu.
   
   The monitoring templates assigned to that computer system are reapplied to that system.

### Applying Monitors to Systems

Typically you will want to create monitoring templates and assign them to groups in order to streamline monitor configuration. However, you can also apply a monitor to an individual computer system. You might want to do this if you want to apply a monitor that is not included in the monitoring template for the group(s) the system is a member of. Or, you may want to do this if you have a system that you want to apply only one monitor to.

Applying a monitor to an individual system is different than a monitor override, which allows you to apply different monitor settings than those specified in the monitoring template. With a monitor override, the monitor is included in the monitoring template, but the configuration settings are altered for the specified system.

Monitors applied to individual systems always take precedence over other monitors. If the same monitor is also included in a monitoring template for a group that the system is a member of, the configuration settings of the monitor applied to the system individually are used and the monitor settings in the monitoring template are ignored.

If the individual system monitor is deleted, the monitor settings from the monitoring template become the active settings.

**Note:** You must have the **USM Modify Individual Monitors for Computer Systems** permission in order to apply a monitor directly to a system (without a monitoring template).

### Apply a Monitor to a System

You can apply a monitor to an individual computer system.

**Follow these steps:**

1. **Do one** of the following:
   - In the tree view, click the system you want to apply a monitor to.
   - In a badge view, drill down to the system you want to apply a monitor to.

2. **Click** the **Monitoring** tab.

3. **Click** **Apply Monitors** in the bottom right of the Monitoring pane.
   
   The **Apply Monitors** dialog is displayed.

4. **(Optional)** To filter the list so you see only a certain category of monitors, choose a category from the **Filter** pull-down menu.
5. Click the Add monitor icon (.addButton).
   The Add Monitor dialog is displayed.
6. Click the monitor you want to add to select it.
   To filter the list so you see only a certain category of monitors, choose a category
   from the Category pull-down menu.
   If you do not see the monitor you want to apply in the list, verify that the probe
   that contains that monitor is installed and activated on the robot that manages the
   computer system.
7. Click OK.
   The configuration dialog for the monitor is displayed.
8. Enter configuration settings for the monitor.
   For information about monitor configuration settings, see the SOC Probe
   Configuration Reference (SOC-probes/index.htm).
9. Click OK.
   The monitor appears in the list in the Apply Monitors dialog.
10. Click Close to dismiss the Apply Monitors dialog.
    It may take several minutes for the monitor to be added to the table.

Delete a Monitor from a System

You can delete a monitor that was applied to an individual computer system.

Follow these steps:
1. Do one of the following:
   ■ In the tree view, click the system you want to delete a monitor for.
   ■ In a badge view, drill down to the system you want to delete a monitor for.
2. Click the Monitoring tab.
3. Click Apply Monitors in the bottom right of the Monitoring pane.
   The Apply Monitors dialog is displayed.
4. (Optional) To filter the list so you see only a certain category of monitors, choose a
   category from the Filter pull-down menu.
5. Click the row of the monitor in the Apply Monitors dialog to select it.
6. Click the Delete monitor icon (deleteButton).
Edit a Monitor for a System

You can edit a monitor that was applied to an individual computer system. Do this if you want to change the configuration settings for the monitor.

Follow these steps:
1. Do one of the following:
   - In the tree view, click the system you want to edit a monitor for.
   - In a badge view, drill down to the system you want to edit a monitor for.
2. Click the Monitoring tab.
3. Click Apply Monitors in the bottom right of the Monitoring pane.
   The Apply Monitors dialog is displayed.
4. (Optional) To filter the list so you see only a certain category of monitors, choose a category from the Filter pull-down menu.
5. Click on the name of the monitor in the Apply Monitors dialog.
   The configuration dialog for the monitor is displayed.
6. Edit the settings, then click OK.
7. Click Close in the Apply Monitors dialog.

Set a Monitor Override

You can set monitor properties to be applied to a specific computer system that are different than the properties configured in the assigned monitoring template. This is called a monitor override.

A monitor override is different than applying a monitor to an individual system. Applying a monitor to an individual system allows you to assign a monitor that is not included in the monitoring template.

Follow these steps:
1. In the tree, expand the group node that the system belongs to.
2. Click on the computer system.
3. Click the Monitoring tab in the right pane.
4. Click the <n> link in the Overrides column for the monitor you want to override.
5. In the Configure Monitor Overrides dialog, click Add.
6. In the Key field, choose the property from the pull-down menu that you want to override.
7. Enter a value in the Value field, then click OK.
View Monitor Status

You can view information about the monitors assigned to a computer system.

Follow these steps:

1. Do one of the following:
   - In the tree view, click the system you want to view monitoring information for.
   - In a badge view, drill down to the system you want to view monitoring information for.

2. Click the **Monitoring** tab.

You see a table that contains this information:

<table>
<thead>
<tr>
<th>Name</th>
<th>Group</th>
<th>Last Update</th>
<th>Template</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name of the monitor assigned to the system.</td>
<td>Group the monitoring template is assigned to.</td>
<td>Date and time the monitor properties were last updated.</td>
<td>Monitoring template that the monitor is associated with.</td>
</tr>
</tbody>
</table>
Overrides

Number of monitor overrides. This number is also a link to the Configure Monitor Overrides dialog. For more information, see Set a Monitor Override (see page 468).

Working with Alarms

Administrators and other authorized users can view and manage alarms in USM.

You can view alarms for a group or for an individual system. Alarm data is summarized in a bar chart, and alarms can be displayed in a list or a table. You can filter alarms or change the way the data is sorted. You can also view a graph of the associated performance metric, if there is one, and view alarm history.

Authorized users can assign and acknowledge alarms. Administrators can set alarms to invisible so they are hidden from other users, and can view invisible alarms.

To support multi-tenancy, you can see and can take action only on alarms from your origin. Authorized users can edit the Access Control List (ACL) to further restrict which alarms users can see. See the section Editing Alarm Filter Dialog (see page 28).

Note: You must have the Acknowledge, Accept, Assign, or Unassign permissions set in the ACL in order to take those actions on alarms. You must have the Invisible Alarms permission in order to set alarms as invisible and to view invisible alarms.

View Alarms

You can view alarms for a group or for a system in the USM Alarms tab. The Alarms tab includes a summary chart and a list view or table view that you can toggle between.

To access the Alarms tab, follow these steps:
1. In the USM tree view, click on the name of the group or system you want to view alarms for.
2. Click the Alarms tab.

Note: Some alarms are not associated with a group or a system. These alarms are displayed under the Groups node in USM.
Alarm Summary Chart

The alarm summary chart displays a bar graph of alarm data for the selected group or system.

By default, the data is grouped by severity. To change the grouping, choose an item from the pull-down menu at the top of the summary chart.

Click on a column in the summary chart to display only that data in the list or table view. To return to viewing all data for the group or system, do one of the following:

- Click the column again.
- Click the X next to the name of the filter in the toolbar above the alarm list or alarm table.
- Choose Clear Filters from the Actions menu. Locked filters are not removed by choosing this menu option.

Click the arrow in the upper right corner of the summary chart to hide the chart; click it again to display the chart.

Alarm List

The alarm list displays details about alarms for the selected group or system. Click the List icon ( ) in the Alarms tab to access the alarm list.

Hover over an item to see what type of information the item contains. For example, the first line contains the Alarm Status icon, Member Details icon, host name, subsystem, origin, and update time. Hover over the alarm message to see the entire message. If a URL is present in an alarm message or in a custom field, you can click the URL to launch it in a new browser window.

Click the following items to filter by that value in the alarm summary chart and the list:

- Alarm Status icon
- Host
- Subsystem
- Origin
- Owner

Click the item again to clear the filter. Or, click the lock icon ( ) next to the filter name to retain the filter when you view alarms for other groups or systems.
Click the **MORE** button to access more items you can click on to filter the data:

- Hub
- Robot
- Source
- Probe
- Assigned By
- User Tag 1
- User Tag 2
- Custom fields (by default named Custom 1 through Custom 5)

Click the Member Details icon, displayed as the icon for the device, to display the device view for the system. To return to the alarm list, click an Alarm Status icon in the device view.

Click the check box next to an alarm to select the alarm. If at least one alarm is selected, the **Actions** menu is activated. You can also choose **Select All** and **Clear Selected** from the **Actions** menu to select or deselect all alarms.

Click the following icons:

- Acknowledge icon (✔️) to acknowledge an alarm.
- Set visible/Set invisible icons (☐/☐) to set whether the alarm is hidden.
- Assign icon (✍️) to change who owns the alarm.
- Add annotation (✍️) to the alarm. Annotations can only be added; previous annotations cannot be edited or deleted. Each entry in the Alarm Annotations editor is time stamped and tagged for the user. HTTP or HTTPS addresses become active links when the annotation is saved.

  **Note:** Annotations are automatically added when alarm ownership changes.

Click the **MORE** button to view more information about the alarm in the **Details**, **History**, and **Metric** tabs.

**Details tab**

Displays information about the alarm, such as the hub and robot that issued the alarm, the probe that collected the data, and any User Tags assigned to the robot.

**History tab**

Displays a table listing timestamps, messages, and assignment actions for the alarm.
Metric tab

Displays a graph with data for the performance metric the alarm was generated for, if appropriate. For example, if the alarm was generated because a threshold for CPU usage was exceeded, the graph displays CPU usage data. The alarm occurrence is indicated on the chart by a circle. Data is displayed for the last 24 hours, unless the alarm is more than 24 hours old, in which case data is displayed for 12 hours before and after the alarm time. Click the Show current data icon to view data for the last 24 hours; click the Show alarm data icon to return to 12 hours before and after the alarm.

Click the Pop out icon in the upper right corner of the graph to display the data as a Performance Report in a new window. This allows you to use additional features in Performance Reporter, such as changing the time interval for the data displayed.

If the alarm is not related to performance data, you see a message stating No metric available when you click the Metric tab. For example, alarms stating that a system rebooted or that a process failed to start do not have performance data associated with them.
Set Custom Alarm Fields

Authorized users can enter text in five custom fields for selected alarms, and the text is displayed in the alarm list view. You may want to use these fields to enter information about who to contact when certain alarms occur, instructions on how to resolve the alarms, or general notes about the alarms. Or, the CA Nimsoft alarm_enrichment probe can be used to automatically add information such as device information (serial numbers, for example) or contact information to the custom text fields. For more information about the alarm_enrichment probe, see the documentation for nas in the Library of Nimsoft Probe Guides.

You must have the Alarm Management permission in your Access Control List (ACL) in order to enter text in the custom alarm fields.

By default, in USM these fields are named Custom 1 through Custom 5. Administrators can change the name of these fields in the Raw Configure window for the Nimsoft Alarm Server (nas).

The text you enter in the custom fields is viewed by clicking the More button in the alarm list view. Or, you can add the columns for the custom fields to the alarms table view.

Note: The custom text fields are displayed in the alarms list view only if text has been entered for the field.

Click on a custom field to filter alarms by that field. You can also group information in the alarm summary chart by custom fields by choosing a custom field from the pull-down menu.

Follow these steps:
1. In the tree view, click the name of a group or a system that you want to view alarms for.
2. Click the Alarms tab.
   The alarms are displayed.
3. Select one or more alarms by clicking the check box next to the alarm(s) or by choosing Select All from the Actions menu.
4. Choose Set Custom from the Actions menu.
   The Set Custom dialog is displayed.
5. Select the fields you want to enter a value for, then enter the desired text.
   Note: Existing text for custom fields is not displayed in the Set Custom dialog. Text you enter overwrites any existing text.
6. Click OK.
7. In the alarms list view, click More for one of the alarms you selected. Or, in the alarms table view choose Edit Columns from the Actions menu and select one or more custom fields.

The custom fields you entered text for are now displayed.

**Change Names of Custom Fields**

The default names for the custom text fields for alarms are Custom 1 through Custom 5 in USM. Administrators can change the name of these fields in the Raw Configure window for the Nimsoft Alarm Server (nas).

**Follow these steps:**

1. Use Infrastructure Manager to open the Raw Configure window for the nas probe:
   a. Select the nas probe.
   b. Press the Ctrl key and right-click on the nas probe.
   c. Choose Raw Configure.
2. Add the setup > custom_headers folder, then open the folder.
3. Click New Key.
   The New Key dialog is displayed.
4. Enter the following values:
   a. Key name = custom_1 through custom_5
   b. Value = The name you want as the label for the field
5. Click OK, then click OK again to close the Raw Configure window.
6. Restart the USM webapp in the wasp probe:
   a. In Infrastructure Manager, double-click on the wasp probe to launch its configuration GUI.
   b. Click the webapps tab.
   c. Right-click USM.
   d. Click Stop.
   e. Click Start.

**Delete Text from Custom Fields**

You can delete text from a custom alarm field.

**Follow these steps:**

1. In the Alarms tab, select one or more alarms by clicking the check box next to the alarm(s) or by choosing Select All from the Actions menu.
2. Choose **Set Custom** from the **Actions** menu.
   The Set Custom dialog is displayed.
3. Select the fields you want to remove text for.
4. Leave the text field(s) blank.
   **Note:** Existing text is not displayed in the Set Custom dialog. Leaving it blank removes any existing text.
5. Click **OK**.
6. In the Alarms list view, click **More** for one of the alarms you selected.
   The custom fields you removed text for are no longer displayed.

**Alarm Table**

The alarm table displays information about alarms in a table format for the selected group or system. Click the table icon (▱) in the **Alarms** tab to access the alarm table.

Click the check box next to an alarm to select the alarm. If at least one alarm is selected, the **Actions** menu is activated. You can also choose **Select All** and **Clear Selected** from the **Actions** menu to select or deselect all alarms.

Click the Alarm Status icon to display only alarms of that status in the alarm summary chart and the table. Click the Alarm Status icon again to clear the filter.

Click the Member Details icon, displayed as the icon for the device, to display the device view for the system. To return to the alarm list, click an Alarm Status icon in the device view.

Click a column header to sort the table data by that column.

You can change the columns displayed in the table by choosing **Edit Columns** from the **Actions** menu and selecting the columns you want to display.

**Note:** You can set values for the columns **Custom 1** through **Custom 5** by selecting one or more alarms and choosing **Set Custom** from the **Actions** menu.

**Export Alarms to CSV**

You can export a list of alarms to a .csv file from the list view or table view of a selected group or system.

To export a list of alarms, click on the **Actions** menu above the list or table, and select **Export to CSV**. A dialog opens on your local system, prompting you to select a location to save the .csv file to.
Manage Alarms

You can accept, assign, unassign, or acknowledge (clear) alarms in the USM Alarms tab.

**Note:** You must have the Acknowledge, Accept, Assign, or Unassign permissions set in the Access Control List (ACL) in order to take those actions on alarms.

**Follow these steps:**

1. In the tree view, click the name of a group or a system that you want to manage alarms for.
2. Click the Alarms tab.
   
   The alarms information is displayed. By default, the list view of alarms is displayed. Click the table icon (️) and list icon (️️️) to toggle between the two alarm views.
3. In the list view, do one of the following:
   - Click the Acknowledge icon (️️️️) to acknowledge an alarm.
   - Click the Assign icon (️️️️️️️). In the popup menu, choose Accept, Assign, or Unassign.
   - Click the check box to select one or more alarms, then choose Accept, Assign, Unassign, or Acknowledge from the Actions menu.
   - From the Actions menu, choose Select All, then choose Accept, Assign, Unassign, or Acknowledge.
4. In the table view, do one of the following:
   - Click the check box to select one or more alarms, then choose Accept, Assign, Unassign, or Acknowledge from the Actions menu.
   - From the Actions menu, choose Select All, then choose Accept, Assign, Unassign, or Acknowledge.

Working with Invisible Alarms

Administrators can set whether or not an alarm is visible to other users. You may want to set some alarms to invisible to hide them if they are not relevant to other users.

You can quickly set a single alarm to invisible (or visible) by clicking an icon, or you can select multiple alarms and set them to invisible (or visible) by choosing a menu item.

**Note:** You must have the Invisible Alarms permission in order to set alarms as invisible and to view invisible alarms.
Set a Single Alarm to Invisible

You can quickly set whether a single alarm is visible by clicking an icon in the alarms list view.

**Note:** You must have the **Invisible Alarms** permission in order to set alarms as invisible and to view invisible alarms.

**Follow these steps:**
1. In the tree view, click the name of a group or a system that you want to manage alarms for.
2. Click the **Alarms** tab.
   
   The alarms information is displayed.
3. Click the list view icon (⿱).
4. Click the Set visible/Set invisible icons (⿱/⿱) to set whether an alarm is hidden for users who do not have the **Invisible Alarms** permission.

Set Multiple Alarms to Invisible

Administrators can filter or sort to find certain types of alarms and then set multiple alarms to invisible (or visible).

**Note:** You must have the **Invisible Alarms** permission in order to set alarms as invisible and to view invisible alarms.

There are several tools that allow you to filter and sort visible and invisible alarms.

To see what other users (who do not have the **Invisible Alarms** permission) will see, choose **Hide Invisible** from the **Actions** menu. All invisible alarms are hidden, and the Visibility filter is listed on the toolbar. Choose **Show All** or click the X icon next to the filter to display the invisible alarms again.

To see how many alarms are visible and invisible, in the list view group the alarm summary chart by choosing **Visibility** from the pull-down menu. To see only invisible alarms, click the **Invisible** column in the alarm summary chart.

In the table view, you can add the Visibility column to the table (choose **Edit Columns** from the **Actions** menu). As with all columns in the table, click the **Visibility** header to sort the table by that column.

**Follow these steps:**
1. In the tree view, click the name of a group or a system that you want to manage alarms for.
2. Click the Alarms tab.
   The alarms information is displayed.

3. Click the list view icon ( ).

4. Select the alarms you want to set to invisible. You can do this by:
   - Clicking check boxes next to the alarms
   - Choosing Select All from the Actions menu
   - Clicking a column in the alarm summary chart to filter for alarms in that column, then choosing Select All from the Actions menu
   - Entering text in the Quick Filter field to display only alarms that contain that text, then choosing Select All from the Actions menu

5. Choose Set Invisible from the Actions menu.

Filter Alarms

There are several ways you can filter alarm data to display only the data you want to see.

- Click a column in the alarm summary chart to display only that data in the summary chart and in the list or table. For example, if the chart is sorted by severity, click on the column for major alarms to see only alarms of major severity.

- Enter text in the Quick Filter field to do a full-text search of alarm data. For example, to find alarms for the dashboard_engine probe, enter dashboard_engine in the Quick Filter field and all alarms with "dashboard_engine" in a text field, such as the alarm message, are displayed.

- Click items in the alarm list or table (the Alarm Status icon, host, subsystem, and so on). For example, to view only alarms related to SLA agreements, click a subsystem item that says SLA.
  - Click the Updated time, Started time, or Assigned time to toggle a date filter. See the section Filter Alarms by Date (see page 480).

- You can freeze the current alarm list or table so that new alarm updates are not displayed. Click the Actions menu > Pause Updates. Alarm updates resume when you select Resume Updates, or when you select a different group or member.

You can combine these methods or click multiple items in the alarm list to apply multiple filters. Active filters are listed in the toolbar above the alarm list or alarm table.

You can click the Exclude matching icon ( ) next to the filter name to view alarms that are not equal to the filter criteria. Click the Include matching icon ( ) to view alarms that match the filter criteria.
You can lock a filter by clicking the lock icon (🔒) next to the filter name. You can lock multiple filters, and you can lock some filters and leave others unlocked.

When a filter is locked, the locked filter state is saved to the database, and is retained when you view alarms for a different group or system. The locked filter state persists until you unlock it. Unlocked filters are cleared when the page is refreshed, or when you select a different group or member.

For example, you are viewing alarms for a group named Servers, and you click the Critical column in the alarm summary chart so that you are filtering for only critical alarms. You click the lock icon to lock the Critical filter, then click on a system in the group named db_server. Because the Critical filter is locked, you see only critical alarms for db_server.

There are several ways to clear filters:

- Choose Clear Filters from the Actions menu to remove all unlocked filters. Locked filters are not removed by choosing this menu option.
- Click the X next to the name of the filter in the toolbar above the alarm list or alarm table.
- If you clicked a column in the alarm summary chart, click the column again to remove the filter.
- If you clicked an item in the alarm list or alarm table, click the item again to remove the filter.

### Filter Alarms by Date

You can add a start and end date to the filter criteria by selecting the Updated time, Started time, or Assigned time of an alarm in the alarm list or table. After the start and end date are added to the filter, you can modify them as follows:

- Select the Exclude matching icon (🚫) to view alarms updated outside of the start and end date displayed.
- Select the Include matching icon (:inline) to view alarms updated within the start and end date displayed.
- Select the start date link to change the start date.
- Select the end date link to change the end date.
Sort Alarms

You can change the sorting of alarm data in the alarm list and alarm table.

To change the sorting of data in the alarm list or alarm table, choose an item from the pull-down menu in the toolbar above the alarm list or table. The sorting you choose is retained when you switch between the alarm list and alarm table.

In the table, click a column header to sort by that column. This updates the Sort by pull-down menu and also sorts the list.

Viewing System Information

You can view information about individual computer systems. The System tab displays information about system properties and performance. The Metrics tab displays data for the metrics monitored on the system.

Follow these steps:
1. Do one of the following:
   - In the tree view, click the system you want to view system information for.
   - In a badge view, drill down to the system you want to view system information for.
2. Click the System tab or Metrics tab.

View System Properties

The System tab displays information about system properties, disk usage, interface traffic, and system performance. The information displayed depends on the data available. Items that have no value available for the system are omitted.

The icons in the Alarms section indicate the number of alarms of each severity level for the system. Click an alarm icon to view the related alarms in the Alarms tab.

If disks are monitored on the system, the Disk Usage table is displayed. Disks are monitored if the cdm (local) or rsp (remote) probe, or a storage probe, is installed and configured to monitor disks on the system.

The Disk Usage table contains a bar for each disk indicating percent of disk usage (or, in the case of some storage probes, percent of disk free). Click the bar to view a Performance Reports chart of disk usage data for the past 24 hours.

Note: You can use the time selection buttons on the bottom toolbar to change the time interval for Performance Reports.
The alarm status icon in the Disk Usage table indicates the highest severity alarm for the disk. Click the alarm icon to view related alarms in the Alarms tab.

If interfaces are monitored on the system (the interface_traffic probe is installed and configured), the Interface table is displayed. Click an interface name in the table to view a Performance Report with data about interface traffic for the past 24 hours.

The alarm status icon in the Interface table indicates the highest severity alarm for the interface. Click the alarm icon to view related alarms in the Alarms tab.

If performance and network response are monitored on the system, you see Performance Reports charts in the right pane of the System tab. The charts displayed depend on which probes are deployed and configured to monitor the system. For example, if the cdm (local) or rsp (remote) probe is collecting data, charts with data on CPU and memory usage are displayed. The charts include an icon indicating alarm status for the metric displayed on the chart.

The charts display data for the previous 24 hours. If data is missing for that interval (for example, if the probe is disabled), the chart is not displayed.

If baseline data is available, it is displayed as a dotted stair-step line on the chart. If baseline data is not available, a trend line for the data is displayed. If you hover over a data point, the baseline or trend line value for the data point is included in the tooltip.

More information:

Device Performance Charts Reference (see page 483)
Device Performance Charts Reference

The following table lists the performance charts displayed for each device type.

<table>
<thead>
<tr>
<th>Device Type</th>
<th>Performance Charts</th>
</tr>
</thead>
<tbody>
<tr>
<td>VMware</td>
<td>■ CPU Reserved Capacity</td>
</tr>
<tr>
<td></td>
<td>■ CPU Usage</td>
</tr>
<tr>
<td></td>
<td>■ CPU Overall Usage (pool)</td>
</tr>
<tr>
<td></td>
<td>■ Disk Latency</td>
</tr>
<tr>
<td></td>
<td>■ Disk Read Rate</td>
</tr>
<tr>
<td></td>
<td>■ Disk Write Rate</td>
</tr>
<tr>
<td></td>
<td>■ Memory Balloon</td>
</tr>
<tr>
<td></td>
<td>■ Memory Granted</td>
</tr>
<tr>
<td></td>
<td>■ Memory Reserved Capacity</td>
</tr>
<tr>
<td></td>
<td>■ Memory Usage</td>
</tr>
<tr>
<td></td>
<td>■ Network Data Receive Rate</td>
</tr>
<tr>
<td></td>
<td>■ Network Data Transmit Rate</td>
</tr>
<tr>
<td></td>
<td>■ VM Count</td>
</tr>
<tr>
<td></td>
<td>■ VM Count Active</td>
</tr>
<tr>
<td></td>
<td>■ Datastore Free</td>
</tr>
<tr>
<td></td>
<td>■ Memory Overall Usage</td>
</tr>
<tr>
<td></td>
<td>■ CPU Usage</td>
</tr>
<tr>
<td></td>
<td>■ Guest Disk Free</td>
</tr>
<tr>
<td></td>
<td>■ Guest Memory Usage</td>
</tr>
<tr>
<td></td>
<td>■ Host Memory Usage</td>
</tr>
<tr>
<td></td>
<td>■ Overall CPU Usage</td>
</tr>
<tr>
<td></td>
<td>■ Power State</td>
</tr>
<tr>
<td>Device Type</td>
<td>Performance Charts</td>
</tr>
<tr>
<td>-------------</td>
<td>-------------------------------------</td>
</tr>
<tr>
<td>Network</td>
<td>CPU Last 5 Minutes</td>
</tr>
<tr>
<td></td>
<td>Memory Free</td>
</tr>
<tr>
<td></td>
<td>Big Buffer Misses</td>
</tr>
<tr>
<td></td>
<td>Huge Buffer Misses</td>
</tr>
<tr>
<td></td>
<td>Very Large Buffer Misses</td>
</tr>
<tr>
<td></td>
<td>Large Buffer Misses</td>
</tr>
<tr>
<td></td>
<td>Medium Buffer Misses</td>
</tr>
<tr>
<td></td>
<td>Small Buffer Misses</td>
</tr>
<tr>
<td>Server</td>
<td>Total CPU Load</td>
</tr>
<tr>
<td></td>
<td>Processor Queue Length</td>
</tr>
<tr>
<td></td>
<td>Memory Paging</td>
</tr>
<tr>
<td></td>
<td>Total Memory Used</td>
</tr>
<tr>
<td></td>
<td>Physical Memory Used</td>
</tr>
<tr>
<td></td>
<td>Memory Swap</td>
</tr>
<tr>
<td></td>
<td>Total Memory Used</td>
</tr>
</tbody>
</table>
View Metrics

The **Metrics** tab allows you to see data for all metrics that are monitored on a system. Metrics are listed in a hierarchical tree. Nodes at any level of the tree may have data, and available data is indicated by the arrow icon (↑) next to the node name.

Click a node in the tree to view charts of the last 24 hours of data, if available, for that node.

**Note:** If you want to view data for an interval other than the last 24 hours, click the pop out icon (смотрите выше) in the upper right corner of the graph to display the data in Performance Reports Designer. From there you can use the time selection buttons at the bottom of the chart to change the time interval.

Charts are ordered by descending alarm severity. If baseline data is available, it is displayed as a dotted stair-step line on the chart. If baseline data is not available, a trend line for the data is displayed. If you hover over a data point, the baseline or trend line value for the data point is included in the tooltip.

If there are more than 20 charts, chart summary bars are displayed instead. Chart summary bars display the alarm status, alarm message, and last sample value, and are updated at 1-minute intervals. Up to 100 summary bars are displayed.

Click the show chart icon (смотрите выше), visible when you mouse over the summary bar, to expand the chart. Once you expand a chart you cannot collapse it.

If there is no sample value in the last 24 hours for a metric, it is not displayed as either a chart or a summary bar.

**View Metrics for Virtual Machines**

For virtual machines (VMs), data is grouped under either the **Virtual Machine** or **Hypervisor** nodes in the tree. Under the **Virtual Machine** node, the VM server, physical host, and VMs (guests) are all displayed.

**Note:** VMs, the VM server, and the physical host are all displayed under the **Virtual Machine** node in the **Metrics** tab. It is recommended that you use a naming convention that allows you to distinguish these components.

Click the **Hypervisor** node to view data about the number of VMs installed and active. You may see more VMs listed in this data than appear in the tree under the **Virtual Machine** node since only monitored VMs appear in the tree. Nodes under the **Hypervisor** node display data about CPU, memory, and resource pool usage.
Display Metrics for Custom Probes

If you have a custom probe, you must include functions provided by the Nimsoft SDK so that data from the probe is displayed in USM. Your probe must create:

- Configuration Items (CIs) to represent monitored items (devices, disks, virtual machines, etc.)
- QoS metrics to contain data from the probe
- Devices to represent the local or remote system that is being monitored

Your probe must link these items to the appropriate alarms and QoS metrics.

For more information about the SDK, see the CA Nimsoft Monitor Solution > SDK Products documentation in the Nimsoft Library.
Following are two examples.

**Example Perl Script for Monitoring a Local Directory**

This example creates a CI that represents a directory on the local system where the probe is running and a QoS metric for available space in the directory. The example uses the function `ciOpenLocalDevice` to create the device to represent the local system. If the item you are monitoring is remote from the system where the probe is running, use the function `ciOpenRemoteDevice` to create a device to represent the system.

```perl
#!perl/bin/perl
use lib "perllib/";
use Nimbus::API;

$ENV{'NIM_ROOT'} = '/opt/nimsoft';
nimInit(0);

# NIS2 - Local device example
# This example shows how to create a CI representing
# a directory on the local system, and a metric representing
# the available space on that directory. A device that represents
# the local computer system where this probe is running is
# also created by the ciOpenLocalDevice function.

my $dirName = "/var";
my $dirCiType = "1.11"; # System.Directory
my $dirMetric = "3"; # Directory Space in KB
my $dirSpace = 100; # Measurement - 100 KB remaining
my $hCI = ciOpenLocalDevice($dirCiType, $dirName);

# Send an alarm for the CI and metric ID
my ($rc, $szId) = ciAlarm($hCI, $dirMetric, 3, "$dirName space is low");

# Define a QoS metric for directory space
my $qosName = "QOS_DIRECTORY_SPACE";
my $qosGroup = "QOS_MACHINE";
my $qosDescr = "Directory Space";
my $qosUnit = "Kilobytes";
my $qosUnitAbbr = "KB";
my $qosInterval = 300;
my $qosSource = ""; # Nimsoft SDK will use the local host address
my $rc = nimQoSSendDefinition($qosName, $qosGroup, $qosDescr, $qosUnit, $qosUnitAbbr);
my $hQoS = nimQoSCreate($qosName, $qosSource, $qosInterval);

# Bind the CI to QoS (establish their relationship)
```
my $rc = ciBindQoS($hCI, $hQoS, $dirMetric);

# Send the QoS data point
my $rc = nimQoSSendValue($hQoS, $dirName, $dirSpace);

# Clean up
my $rc = ciUnBindQoS($hQoS);
my $rc = nimQoSFree($hQoS);
my $rc = ciClose($hCI);

nimEnd(0);
Example NimRecorder Script for e2e_appmon Probe

This example, created using the NimRecorder scripting wizard provided with the developer edition of the probe (e2e_appmon_dev), creates CIs that represent each web page that is monitored ("Support" and "Downloads"). The example also sends QoS measurements for the time it took to render each web page.

The NimRecorder is launched by choosing the Programs > Nimsoft Monitoring > E2E Scripting menu option. For more information about using NimRecorder, see the documentation accessed from the Help menu in the NimRecorder application window.

```
include "NimBUS-functions.src"
nimSetCi("3.21", "Support", "", "1")
nimInit()

StartBrowser("IE", "www.nimsoft.com", 3)

target$= "Support"
nimQoSStart()

UsePage("ITSM + Service Desk + IT Monitoring: Better Together - Nimsoft")
  ClickHTMLElement("A[INNERTEXT='support']")

nimQoSStop()

nimQoSSendTimer(target$)

nimSetCi("3.21", "Downloads", "", "1")
nimInit()
target$= "Downloads"
nimQoSStart()

UsePage("ITSM and ITIL-based Support and Service That You Can Count On - Nimsoft")
  ClickHTMLElement("A[INNERTEXT='Downloads',INDEX='2']")

nimQoSStop()

nimQoSSendTimer(target$)

CloseWindow("IEXPLORE.EXE|IEFrame|Nimsoft - Windows Internet Explorer",1)

CloseWindow("IEXPLORE.EXE|IEFrame|ITSM and ITIL-based Support and Service That You Can Count On - Nimsoft - Windows Internet Explorer",1)

nimEnd()
```
Working with Reports

You can assign report templates to groups in USM. After a report template is assigned to a group, users can view the report for any system in the group.

USM comes with a default report template. In addition, there are ready-made report templates with server usage and performance information. You can find these in the Reports Template dialog (see Assigning Report Templates to Groups (see page 490)), in a folder named USM Dynamic Report Templates. These reports are:

**Server Basic**
- Displays information on total CPU usage, system CPU usage, user CPU usage, idle CPU usage, wait CPU usage, percent of memory used, percent of physical memory used, megabytes of memory used, megabytes of swap memory used, paging memory used, and disk space usage.

**Server Load**
- Displays information on CPU and processor queue length and megabytes of physical and swap memory used.

**Server Performance**
- Displays information on total CPU usage, user and system CPU usage, CPU processor queue length, percent memory used, megabytes of physical and swap memory used, total megabytes of memory used, and disk space usage.

You can also create reports in the Performance Reports Designer and assign them as report templates to groups in USM.

You can designate which report template is the default report. The default report is launched when users click the Reports button. There must be one report template designated as the default, and the default template cannot be deleted.

Assigning Report Templates to Groups

Administrators who have the USM Group Modification permission can assign report templates to groups in USM. Once you assign the report template to a group, users can view the report for any system in the group.

**To assign report templates to a group:**

1. In a badge or tree view, click on a group and choose Edit Group from the Actions menu. Or, in the tree view mouse over the name of a group and click the Edit Group icon ( ).
   - The Edit Group <Group Name> dialog is displayed.
2. Click the Report Templates tab.
3. Click **Add Template**.
   The Report Templates dialog is displayed. If you are an account user, you see only the report templates that are not private to another account or to another user. Otherwise, you see all report templates that are not private to another user.

4. Select the report template you want to add, then click **Add**.
   The report template you added is assigned to the selected group and all groups and systems under it in the tree.

5. (Optional) To select a report template as the default report for the group, click the radio button in the **Default** column for that template.
   The default template is the report launched when users click the **Reports** button in USM. There must be one report template designated as the default, and the default template cannot be deleted.

**Viewing Reports**

**To view a report for a computer system:**

1. Do one of the following:
   - In the tree view, click the system you want to view a report for.
   - In a badge view, drill down to the system you want to view a report for.

2. To view the default report, click the **Reports** button. To select another report, click the arrow next to the **Reports** button and choose the report from the pull-down menu.
   The report data is displayed in a new browser window.

**Maintenance Mode**

The maintenance mode feature lets you temporarily suppress monitoring so that alarms are not generated for systems in maintenance mode. You can schedule recurring maintenance to perform routine system updates. Or, you can place systems in a schedule that runs once only.

You can create an ad hoc maintenance schedule if an unplanned outage occurs. You can quickly place a system in maintenance mode while you respond to the outage.

To manage maintenance schedules, you must have the **Edit Maintenance Mode Schedules** permission set in the NMS Access Control List (ACL). To manage the systems in maintenance schedules, you must have the **Edit Maintenance Mode Devices** permission set in the NMS ACL.
Create or Edit a Maintenance Schedule

A maintenance schedule must be created before you can add systems to the schedule.

Follow these steps:
1. Select a group in the USM navigation tree, then select the Maintenance tab.
2. Click the New Schedule button. Or, to edit an existing schedule, select the gear icon (⚙️) next to the schedule you want to edit.
   The Maintenance Schedule window opens.
3. In the Maintenance Schedule window:
   a. Enter a name, or update the existing name. If desired, enter a description.
   b. Select an account for the maintenance schedule if desired.
   c. Make a selection under the Scheduling heading.
      The Schedule Recurrence options to the right update dynamically based on your selection.
   d. Select from the options under Schedule Recurrence and Maintenance Window as desired.

The new schedule is displayed, and you can add groups and systems.

Add Systems to a Maintenance Schedule

After a maintenance schedule is created, you can add groups or systems to the schedule.

Note: Groups and systems can be added to multiple maintenance schedules.

Follow these steps:
1. Browse to a group in the USM navigation tree, and select the Maintenance tab.
   Note: You can expand groups to display individual systems, but the Maintenance tab is only displayed when a group is selected in the navigation tree.
2. Hover over the handle icon (¶¶¶) next to a group or system, and use the grabber icon ( dataSnapshot IMG ) to drag-and-drop systems to a maintenance schedule.
   You can also use the Search field to find systems that do not belong to a group. Drag systems from the Search Results node in the navigation tree.

The maintenance schedule is updated and lists the system or group that you added. The wrench icons (🔧 - inactive; 🔧 - active) indicate that a group or system belongs to at least one maintenance schedule.
Perform Ad Hoc Maintenance

You can create an *ad hoc* maintenance schedule from the context of an individual system, and the schedule is prepopulated with the system. This allows you to quickly respond to the failure. You can drag-and-drop more systems into the schedule if desired.

**Follow these steps:**

1. Browse the USM navigation tree and select a system.
   The System tab is displayed.

2. Click the icon in the upper right-hand corner.
   The Maintenance Schedule dialog opens. The Name field prepopulated with the system you selected.

3. Update the options in the Maintenance Schedule window as desired.

After you click Save, you can drag-and-drop more systems to the maintenance schedule if desired.
Chapter 26: Managing Nimsoft Monitor Health

The state of your monitoring infrastructure is indicated by the Nimsoft Monitor Health icon in the right pane of Unified Service Manager (USM). The green heart icon (💖) indicates that all monitoring components are in a healthy state. The broken heart icon (💔) indicates that one or more monitoring components are in a failed state.

Clicking a Nimsoft Monitor Health icon (💖 or 💔) opens a window in which you can restart failed components or view log files. If necessary, you can restart the entire system.

Follow these steps to manage your monitoring infrastructure:

1. In the right pane of USM, click the Nimsoft Monitor Health icon (❤️).
   The Nimsoft Monitor Health window opens.
2. Locate any components that are in a failed state (🚫).
3. Click the Restart icon (🔄) next to a failed component. You can also click the View log icon (🔧) next to a component.
   If multiple components are in a failed state, you can select Actions > Attempt to Fix. This option restarts all components that are in a failed state.
4. If one or more components are still in a failed state after being restarted, restart the entire system by selecting Actions > Restart.
5. If a component is deactivated (.onPause), you can activate it by clicking the play button (▶).

Launching a Standalone USM

You can launch USM in a browser window as a standalone application (outside of UMP). You may want to do this if you want to link to USM from another application, or if you want to display data for a group or system in USM on a web page.

The standalone USM opens in a new browser window. Because it is a standalone version of USM, you cannot navigate to other UMP portlets or change UMP settings from this window.
There are two ways to launch a standalone USM window: using a URL, and using an HTML file. Using a URL is simple and flexible. Using an HTML file allows you to hide the user name and password or to incorporate USM in another web page (displaying it in a frame, for example).

**Launching USM with a URL**

You can launch a standalone version of USM using a URL with the following syntax:

```
http://<host name or IP address>/usm/jsp/standalone.jsp?user=<user name>&password=<user password>
```

This displays USM in the tree view with the root **Group** node selected.

You can omit the user name and password from the URL. In this case, the user is prompted for a user name and password when the URL is executed.

**Specify a Group OR a System**

You can specify a group or computer system to display by appending the parameters described in the following table to the URL. USM searches from the top of the tree and displays the first instance of the group or system name in the tree.

For group or system names that contain a space, you can enter either a space or a + in the URL. The parameters are not case-sensitive.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Example(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>group</td>
<td>Name of the group you want to display. Enter the full name of the group.</td>
<td>ump_server/usm/jsp/standalone.jsp?user=admin&amp;password=myump&amp;group=acme</td>
</tr>
<tr>
<td>grp_id</td>
<td>ID number of group from the cm_group table. Obtain the ID number by querying your Nimsoft database.</td>
<td>ump_server/usm/jsp/standalone.jsp?user=admin&amp;password=myump&amp;grp_id=530</td>
</tr>
</tbody>
</table>
| device    | Host name or IP address. If you specify the fully-qualified host name, the first instance of the host is displayed. If you enter part of the host name, the first instance of a host that contains that text in the name is displayed. If the system you specify exists but does not belong to a group, the standalone USM displays it under the Search node of the tree. | ump_server/usm/jsp/standalone.jsp?user=admin&password=myump&device=server1.acme.com  
ump_server/usm/jsp/standalone.jsp?user=admin&password=myump&device=172.24.135.16  
ump_server/usm/jsp/standalone.jsp?user=admin&password=myump&device=server1  |
### Parameter | Value | Example(s)
--- | --- | ---
*cs_id* | ID number of the computer system from the cm_computer_system table. Obtain the ID number by querying your Nimsoft database. | `ump_server/usm/jsp/standalone.jsp?user=admin&password=myump&cs_id=271`

*view* | USM view to display. If no view is specified, the standalone USM opens in the tree view. Possible values are:
- badge or badge 1 - Displays USM in the badge 1 view
- badge2 - Displays USM in the badge 2 view
- badge3 - Displays USM in the badge 3 view
- alarms - Displays USM with the **Alarms** tab displayed | `ump_server/usm/jsp/standalone.jsp?user=admin&password=myump&group=acme&view=badge2`

### Specify a Group AND a System
You can specify both a group and a system by using the group or grp_id parameter and the device or cs_id parameter. For example:

`ump_server/usm/jsp/standalone.jsp?user=admin&password=myump&group=acme&cs_id=271`

### Specify a Path
You can display a specific group or system, rather than having UMP search from the top of the tree, by including the path to the group or system in the USM tree structure.

Begin the path with a slash; you can use either forward slashes (/) or back slashes (\). If you have a group with a forward slash in the name, use back slashes to specify the path, and vice versa. Do not include the root **Group** node in the path.

For example:
- `ump_server/usm/jsp/standalone.jsp?user=admin&password=myump&group=\customers\acme`
- `ump_server/usm/jsp/standalone.jsp?user=admin&password=myump&device=/customers/acme/server1`
**Which Item is Displayed?**

This topic looks at some examples to illustrate how the URL syntax works. We have the following tree structure.

![Tree Structure](image)

The following table lists values for the group parameter and describes the results for each value.

<table>
<thead>
<tr>
<th>Group Value</th>
<th>Group Selected</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>group=test</td>
<td><img src="image" alt="Test Group" /></td>
<td>The first group in the tree named Test is selected.</td>
</tr>
<tr>
<td>group=/test/test</td>
<td><img src="image" alt="Test Group" /></td>
<td>The initial slash indicates a path, so the Test subgroup is selected.</td>
</tr>
<tr>
<td>group=test/test</td>
<td><img src="image" alt="Test Group" /></td>
<td>Without an initial slash the text is treated as a group name, not a path, and the Test/Test group is selected.</td>
</tr>
</tbody>
</table>

**Errors**

This section describes error messages you may see when launching a standalone USM with a URL.

**System Not Found**

The member <system name or cs_id> was not found.

**Reason:**
The specified system name or cs_id does not exist.

**Action:**
Enter a valid system name or ID.
Group Not Found

The group <group name> was not found.

Reason:
The specified group name or grp_id does not exist.

Action:
Enter a valid group name or ID.

Launching USM using an HTML File

You can use an HTML file to launch the standalone USM. You may want to do this if:

- You do not want to pass parameters, such as user name and password, in a URL
- You want to display the standalone USM as part of an HTML page, such as in a frame

Follow these steps:

1. Create an HTML file with the following syntax:

```html
<html>
<body onload="javascript:document.autoForm.submit()">
<form name="autoForm" method="post" action="http://<UMP SERVER>/usm/jsp/standalone.jsp">
  <input type="hidden" name="group" value="<GROUP NAME>">
  <input type="hidden" name="device" value="<DEVICE NAME OR IP>">
  <input type="hidden" name="user" value="<USER NAME>">
  <input type="hidden" name="password" value="<USER PASSWORD>">
</form>
</body>
</html>
```

2. Execute the HTML file.

For example, enter the path to the file as a URL in a browser:

http://<SERVER>/<PATH TO HTML FILE>

The standalone USM opens in a browser window without displaying parameters in the URL.
3. (Optional) To display the standalone USM as part of an HTML page, refer to the HTML file created in step 1 on your HTML page.

For example, to display USM in an iframe, include syntax similar to the following in your HTML page:

```html
<html>
  <body>
    <iframe src="http://<SERVER>/<PATH TO HTML FILE>" width="100%" height="300" frameborder="0">
    </iframe>
  </body>
</html>
```
Chapter 27: Unified Dashboards Guide


This document contains information about:

- How to update dashboards for existing users
- Which probes and QoS must be enabled to collect data for each dashboard
- A description of each dashboard
Chapter 28: Administration

This section contains the following topics:

- Single Sign On (see page 503)
- Probe Updates (see page 504)
- Set Permissions for UMP Portlets (see page 504)

Single Sign On

UMP supports SAML (Security Assertion Markup Language) 2.0-compliant single sign-on. Single sign-on allows users, once logged into your systems, to access UMP without logging in again.

SAML is an XML-based open standard for exchanging authentication and authorization data between security domains, that is, between an identity provider (a producer of assertions) and a service provider (a consumer of assertions).* Support for SAML-based authentication is available in a Software-as-a-Service (SaaS) model from a third-party company, AssureBridge.

In order to use the AssureBridge SAML single sign-on solution for UMP, you must have an identity server with a public URL.

For more information or to obtain single sign-on, contact AssureBridge:

- Email: info@assurebridge.com
- Phone: 1-888-409-6995
- Visit www.assurebridge.com

*Source: http://en.wikipedia.org/wiki/Saml
**Probe Updates**

Probes in the CA Nimsoft Archive are automatically deployed to CA Nimsoft robots. However, if an updated version of the probe is placed in the Archive, the updated version of the probe is not deployed to robots where an earlier version of the probe is already installed.

An upcoming version of the product will manage version updates of probes. In the meantime, if you need to update the version of a probe installed on a robot you can do it manually using Infrastructure Manager. To update a probe using Infrastructure Manager, drag the probe package from the archive list and drop it on a robot node.

---

**Set Permissions for UMP Portlets**

To access UMP portlets, users must have the appropriate permissions set in the NMS Access Control List (ACL). ACL permissions are set in the Account Admin portlet or in Infrastructure Manager. A "permission denied" message is displayed when users try to access a portlet for which they do not have the required permission.

The following portlets do not allow access to account contact users, regardless of permissions set:

- Nimsoft Remote Admin
- SLM

The following table describes the permissions for UMP portlets.

<table>
<thead>
<tr>
<th>Portlet</th>
<th>Required Permission to Access Portlet</th>
<th>Other Available Permissions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Account Admin</td>
<td>Account Administration</td>
<td>Manage ACL - create, modify, and delete ACLs.</td>
</tr>
<tr>
<td>Alarm Console</td>
<td>Alarm Details</td>
<td>Alarm action permissions:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ Accept</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ Unassign</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ Invisible Alarms</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ Alarm History</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ Acknowledge</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ Alarm Management</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ Assign</td>
</tr>
<tr>
<td></td>
<td></td>
<td>■ Reassign</td>
</tr>
<tr>
<td>Change Password</td>
<td>Change Password</td>
<td></td>
</tr>
<tr>
<td>-----------------</td>
<td>-----------------</td>
<td></td>
</tr>
<tr>
<td><strong>Note:</strong> In addition to having the Change Password permission set in the ACL, the user must be an account contact user in order to access this portlet.</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Cloud Monitor</th>
<th>Cloud UE Monitor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Custom Dashboards <strong>Note:</strong> This portlet is replaced in UMP 7.5 by the Dashboard portlet. However, existing customers still have access to this portlet.</td>
<td>Custom Dashboards</td>
</tr>
</tbody>
</table>

| Dashboard Designer **Note:** This portlet is replaced in UMP 7.5 by the Dashboard portlet. However, existing customers still have access to this portlet. | Dashboard Designer - allows the user to access the portlet  
Dashboard Publish - allows the user to publish dashboards |
|-----------------|---------------------------------------------------|
| Dashboard       | Dashboard Designer - allows regular Nimsoft users to create, edit, and publish dashboards  
Custom Dashboards - allows account contact users to view dashboards |

<table>
<thead>
<tr>
<th>Discovery Status</th>
<th>Discovery Pie</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Set Permissions for UMP Portlets

<table>
<thead>
<tr>
<th>Dynamic Views</th>
<th>Dynamic Views Dashboards</th>
</tr>
</thead>
<tbody>
<tr>
<td>List Designer</td>
<td>List Designer</td>
</tr>
<tr>
<td>List Viewer</td>
<td>List Viewer</td>
</tr>
<tr>
<td>My Tickets</td>
<td>Service Desk</td>
</tr>
<tr>
<td>Nimsoft Remote Admin</td>
<td>Basic Management</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> The Basic</td>
</tr>
<tr>
<td></td>
<td>Management permission</td>
</tr>
<tr>
<td></td>
<td>allows users to take actions</td>
</tr>
<tr>
<td></td>
<td>in other CA Nimsoft</td>
</tr>
<tr>
<td></td>
<td>Monitor applications, such</td>
</tr>
<tr>
<td></td>
<td>as starting and stopping</td>
</tr>
<tr>
<td></td>
<td>probes in Infrastructure</td>
</tr>
<tr>
<td></td>
<td>Manager.</td>
</tr>
<tr>
<td>NetFlow</td>
<td>NetFlow</td>
</tr>
<tr>
<td>Performance Reports Designer</td>
<td>QoS Access</td>
</tr>
<tr>
<td>Reports</td>
<td>Custom Reports</td>
</tr>
<tr>
<td>Report Scheduler</td>
<td>Report Scheduler</td>
</tr>
<tr>
<td>Service Desk</td>
<td>Service Desk</td>
</tr>
<tr>
<td>SLA Reports</td>
<td>SLM View</td>
</tr>
<tr>
<td>SLM</td>
<td>SLM Admin</td>
</tr>
<tr>
<td>Unified Reporter</td>
<td>Unified Reports</td>
</tr>
</tbody>
</table>
### Unified Service Manager

**USM Basic or Basic Management**

**Note:** The Basic Management permission allows users to take actions in other CA Nimsoft Monitor applications, such as starting and stopping probes in Infrastructure Manager. Use the USM Basic permission to grant USM access while restricting access to other areas of CA Nimsoft Monitor.

- USM Edit Monitoring Templates
- USM Group Modification
- USM Automatic Robot Installation
- USM Modify Individual Monitors for Computer Systems
- Edit Maintenance Mode Devices
- Edit Maintenance Mode Schedules
- Alarm Management - enter text for alarms in five custom fields (by default named Custom 1 through Custom 5).
- Invisible Alarms - see invisible alarms and set alarms to be invisible.

**Alarm action permissions:**

- Accept
- Acknowledge (clear)
- Assign
- Unassign
Chapter 29: Troubleshooting

This section contains the following topics:

- Single Time Zone Required (see page 509)
- LDAP Authentication with Active Directory Groups (see page 510)
- Custom Logo Missing (see page 510)
- Unified Service Manager Displays Non-localized Text (see page 510)
- Relationship Services Not Supported on Solaris (see page 511)
- I Cannot Access a Wiki Page I Created (see page 511)
- Page Titles May Contain Garbled Text (see page 512)
- Garbled Text in Console Installer (see page 512)
- Relationship Viewer Dependent upon Root Cause Analysis and Topology Manager (see page 513)
- Close Browser When Logging Out (see page 513)
- Size of Dashboard Panels (see page 513)
- Do Not Edit QoS Properties in SLM Portlet (see page 513)
- Turn Off Compatibility View with IE 8 or 9 (see page 514)
- Internet Explorer 8 Performance Issue (see page 514)
- Internet Explorer 8 Displays Page Title Incorrectly (see page 514)
- Portlet Not Resizing in Chrome (see page 515)
- User Names (see page 515)
- Internet Explorer 8 Cross-Site Scripting Issue (see page 516)
- Login Fails on Solaris (see page 517)
- Slow Performance on Internet Explorer 7/8 (see page 517)
- Non-ASCII Characters Stored Incorrectly in the Database (see page 517)
- Numeric Login IDs Do Not Work (see page 518)
- I Cannot Access the Control Panel (see page 518)
- Logging In Takes Me to the Login Page (see page 519)
- CSV File is Incorrect in Chinese or Japanese (see page 519)

Single Time Zone Required

For data time-stamping to work correctly across a distributed Nimsoft deployment, the Nimsoft Server, the UMP server, and the database server must all be set to the same time zone, regardless of the geographic locations of the servers.
LDAP Authentication with Active Directory Groups

In the Account Admin portlet, you can link a Nimsoft ACL to an Active Directory group so that users can authenticate using LDAP. However, a user’s primary group in Active Directory cannot be the same as the LDAP group you select for the Nimsoft ACL.

Custom Logo Missing

Symptom:
After I upgraded to UMP 2.5.x, my custom logo is missing.

Solution:
This is due to a change in the behavior of the third-party portal software, Liferay 6. In Liferay 5 custom images were stored in the database. In Liferay 6 custom images are stored on the file system, and Liferay cannot find existing custom images in the database. To resolve this, specify the custom image for the logo again.

Do the following steps:
1. Log into UMP as an administrator.
2. Click Manage, Control Panel from the menu bar.
3. Click Settings under Portal.
4. Click Display Settings under Miscellaneous on the right.
5. Click Change under Logo and browse to the image you want to use for the logo.
   A small .jpg file works best.
6. Click Save.
   Your pages now display the logo you specified.

You may also need to specify other custom images, such as for a user or organization, again.

Unified Service Manager Displays Non-localized Text

Valid after upgrading from NMS 5.12 to NMS 5.60 with non-English locales

In the Unified Service Manager portlet, if you select the Windows or UNIX server group in the left-hand navigation pane, the Description field in the right-hand pane displays English text, instead of the localized language.
Relationship Services Not Supported on Solaris

The Relationship Viewer portlet requires the IP address and port of the Graph Services within the relationship_services probe.

However, in this release of UMP, the relationship_services probe does not run on a Solaris hub. So if you run the installer from a Solaris primary hub, you are not presented with the option to specify the relationship_services probe location.

However, the relationship_services probe does not have to be on the primary hub, and it may actually be present somewhere else in your environment.

The installer will try to find a robot that is running the relationship_services probe. If that search is successful, the installer will use it to configure the Relationship Viewer portlet. If the search fails, the installer will not attempt to configure the Relationship Viewer portlet.

I Cannot Access a Wiki Page I Created

Symptom:
I create a page for a global wiki, but when I try to view the page I see this error: You do not have the required permissions.

Solution:
To resolve this, the owner of the wiki must grant Power User permissions for the wiki page.

Follow these steps:

Note: These steps must be done by the owner of the global wiki (the user who created the wiki).

1. Click Manage, Control Panel.
   The Control Panel is displayed.
2. Click Wiki under My Community.
3. Click the name of the wiki.
4. In the All Pages table, click Actions, Permissions for the page you want to allow access to.
   The permissions dialog for the wiki page is displayed.
5. Select all the check boxes for Power User, then click Save.
   All users can now access the wiki page.
Page Titles May Contain Garbled Text

Valid for non-English instances of UMP

Symptom:
I imported a LAR file, and now page titles in UMP are not displaying correctly.

Solution:
If this occurs, you may need to add an additional Java startup option to the wasp configuration to set the encoding to UTF-8.

Follow these steps:
1. In Infrastructure Manager, double-click the wasp probe.
2. Click the Setup tab.
3. At the bottom of the window, under the heading Extra Java VM arguments, enter the following text:
   -Dfile.encoding=UTF-8
4. Click OK.
5. Deactivate the wasp.
6. Activate the wasp.

Garbled Text in Console Installer

Valid on Windows systems

Symptom:
I see garbled characters when I run the installer in console mode for the Spanish or Brazilian Portuguese version of UMP.

Solution:
This is due to a defect in the third-party installation tool. We have requested a fix for this issue. In the meantime, the workaround is to run the installer in its default mode, which displays a graphical user interface (GUI).
Relationship Viewer Dependent upon Root Cause Analysis and Topology Manager

At present, the only source of data for the Relationship Viewer is the CA Nimsoft Root Cause Analysis and Topology Manager. Attempting to use Relationship Viewer without it generates an error message stating that there is no data to display. Contact CA Nimsoft sales for more information about Root Cause Analysis and Topology Manager.

Close Browser When Logging Out

When you log out of UMP it is important to also close the browser. This fully terminates active sessions. Otherwise, you may see unexpected behavior.

Size of Dashboard Panels

The size of the data returned for a dashboard panel should not exceed 5 MB. If it does you may observe either a slow response or scrollbars freezing or not scrolling the content in the dashboard. To estimate the data sent for a panel, use the following guidelines:

- Gauge, meter, slider: on average 50 bytes
- Chart: on average 50 bytes for each sample
- Table: size of data returned from query + overhead for each cell equal to (the length of the column name x 2) + 5 bytes

If the data for a panel is too large UMP logs a message and sends notification to the requesting client.

Do Not Edit QoS Properties in SLM Portlet

In the SLM portlet, you can expand the Quality of Service pane and view the various QoS in the database. If you open the Properties dialog for a QoS Definition, the fields in the dialog are editable. However, if you edit these fields, the data in the S_QOS_DEFINITION database table changes and conflicts with other database tables. This causes unwanted behavior in other portlets and with other functionality.

The only supported way to change QoS definitions is in the interface of QoS-enabled probes.

Important! Do not edit the fields in the Properties dialog of a QoS in the SLM portlet.
Turn Off Compatibility View with IE 8 or 9

If you use Microsoft Internet Explorer 8 or 9 to access UMP, you must turn off the Compatibility View for navigation menus to work properly. Check the Compatibility View Settings under the Tools menu in Internet Explorer to be sure Compatibility View is turned off for UMP.

Internet Explorer 8 Performance Issue

The 64-bit versions of Internet Explorer 8 and 9 do not work with UMP in a stable and consistent manner on all platforms of Windows. Currently only the 32-bit versions of Internet Explorer 8 and 9 are supported for use with UMP.

Internet Explorer 8 Displays Page Title Incorrectly

If you open certain portlets with Internet Explorer 8, and then click within the portlet, the pound or hash symbol (#) is displayed in the browser tab, rather than the portlet name.

This may occur with the following portlets:

- PRD
- SLM
- List Designer
- Custom Dashboards
- Unified Services Manager
- SQL Server Load (My SQL Server)
- Exchange
- Power
- Vblock
- Network
- Unified Reports
Portlet Not Resizing in Chrome

Valid for Google Chrome

Symptom:
When I resize my browser window, the portlet I'm using becomes unresponsive. If I try to close the window, a message referencing a memory error appears.

Solution:
A number of portlets may become unresponsive if you resize the browser window while using Google Chrome. This is due to an issue with Flash player in Chrome that can occur if you make the browser window very small, and then increase its size again.

If you experience this issue, refresh the browser to make the portlet display correctly again, and then avoid resizing the window while using Chrome.

User Names

CA Nimsoft Account Contact login IDs and user names must be in all lowercase characters.

Important! Do not use mixed-case names or uppercase names as they will not work properly in UMP.
Internet Explorer 8 Cross-Site Scripting Issue

Valid on Internet Explorer 8 Browser

Symptom:
When I use Internet Explorer 8 to launch a Performance Reports Designer report from a URL, I see an Information triangle that states:

Unable to locate new series.

Data not located for: 10.0.4.222|QOS_INTERFACE_PACKETS|PACKETS_IN-Intel(R) PRO/100 VE Network Connection - Packet Scheduler Miniport.

The IE information bar displays this message:

Internet Explorer has modified this page to prevent cross-site scripting.

Solution:
Usually this means a report template was applied to a system for which not all of the QoS objects were defined or that a QoS object was incorrectly specified in the URL.

To resolve this issue:
1. Choose Tools > Internet Options from the Internet Explorer menu bar.
2. Click the Security tab.
3. Click Trusted Sites.
4. Click Sites.
5. Add the site to the zone.
6. Uncheck Require server verification (https:) for all sites in this zone.
7. Click Close.
8. Click Custom level.
9. Scroll to near the bottom of the list of Settings and for Enable XSS filter click Disable.
10. Click OK to close the dialogs.
Login Fails on Solaris

Symptom:

After I install UMP on Solaris I try to log in as the administrator user and the login fails.

Solution:

Use Infrastructure Manager to log into the hub. This prompts CA Nimsoft Monitor to create the administrator user in the database, and you can now log into UMP as the administrator user.

Slow Performance on Internet Explorer 7/8

Symptom:

You may experience that UMP works slowly using IE7, possibly due to IE7’s caching strategy.

Solution:

You can optimize this by changing the settings for browsing history. Go to Tools-InternetOptions-General-BrowsingHistory-Settings, and set the strategy for checking pages to Automatic.

For IE8 it may also help to select the CompatibilityView mode.

Non-ASCII Characters Stored Incorrectly in the Database

Depending on the database collation used, the database may not store non-ASCII user input correctly. For example, if a user enters Japanese characters in a community string in the Nimsoft Remote Admin portlet, the database may store the Japanese characters incorrectly, causing authentication to fail.
Numeric Login IDs Do Not Work

Symptom:
I cannot log in using a numeric user name.

Solution:
Do the following steps:
1. Deactivate the wasp probe in Infrastructure Manager.
2. Open the following file for editing: <UMP Installation directory>/probes/service/wasp/webapps/ROOT/WEB-INF/classes/portal-ext.properties.
3. Add the following line:
   users.screen.name.allow.numeric=true
4. Activate the wasp probe.

I Cannot Access the Control Panel

Symptom:
After upgrading from UMP 2.1.x (or earlier) to UMP 2.5.x (or later), when I click Manage, Control Panel I see my private home page instead of the Control Panel.

Solution:
Users not assigned to the Liferay administrator role may not be able to access the Control Panel after upgrading.

Do the following steps:
1. Deactivate the wasp probe in Infrastructure Manager.
2. Open the probes/service/wasp/webapps/ROOT/WEB-INF/classes/portal-ext.properties file for editing.
3. Add the following line, or if it is present uncomment it by removing the hash sign character (#):
   permissions.user.check.algorithm=5
4. Activate the wasp probe in Infrastructure Manager.
Logging In Takes Me to the Login Page

Symptom:
After upgrading from UMP 2.1.x (or earlier) to UMP 2.5.x (or later), when I log in I see the UMP login page again instead of my private home page.

Solution:
Do the following steps:
1. Deactivate the wasp probe in Infrastructure Manager.
2. Open the probes/service/wasp/webapps/ROOT/WEB-INF/classes/portal-ext.properties file for editing.
3. Add the following line, or if it is present uncomment it by removing the hash sign character (#):
   virtual.hosts.default.community.name=
4. Activate the wasp probe in Infrastructure Manager.

CSV File is Incorrect in Chinese or Japanese

Symptom:
When I export a Performance Report to a CSV file in Chinese or Japanese, there are incorrect characters and incorrect times in the report.

Solution:
This may occur if the locale in UMP is set to a multi-byte language (Chinese or Japanese) and your default application associated with CSV files is Microsoft Excel. Excel does not always correctly identify the encoding (UTF-8) of the exported CSV file.
To resolve this, use the Excel Import function. The following steps are an example for Excel 2007:
1. Click the Data tab in the top ribbon in Excel.
2. Click From Text in the Get External Data section.
3. Browse to the CSV file and click Import.
4. Click the Delimited radio button for the Original data type.
5. Choose 65001: Unicode (UFT-8) from the File Origin pulldown menu.
6. Click Next, then click the Comma check box in the Delimiters section.
   Make sure only the Comma check box is checked.
7. Click Next, then click Finish.
# Index

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>$ PASSWORD Substitution in Alarm Console</td>
<td>56</td>
</tr>
<tr>
<td>Access to USM</td>
<td>402</td>
</tr>
<tr>
<td>Accessing the Relationship Viewer</td>
<td>254</td>
</tr>
<tr>
<td>Account Admin</td>
<td>25</td>
</tr>
<tr>
<td>Account Admin Pane</td>
<td>26</td>
</tr>
<tr>
<td>Account Tables</td>
<td>376</td>
</tr>
<tr>
<td>Actions Dialog</td>
<td>45</td>
</tr>
<tr>
<td>Add a Border</td>
<td>97</td>
</tr>
<tr>
<td>Add a Database for SQL Data Sources</td>
<td>88</td>
</tr>
<tr>
<td>Add a Gauge</td>
<td>77</td>
</tr>
<tr>
<td>Add a Line</td>
<td>66</td>
</tr>
<tr>
<td>Add a Line Chart</td>
<td>80</td>
</tr>
<tr>
<td>Add a Linear Gauge</td>
<td>78</td>
</tr>
<tr>
<td>Add a Note to a Ticket (Self Service Users)</td>
<td>224</td>
</tr>
<tr>
<td>Add a Page</td>
<td>20</td>
</tr>
<tr>
<td>Add a Shadow</td>
<td>97</td>
</tr>
<tr>
<td>Add a Shape</td>
<td>66</td>
</tr>
<tr>
<td>Add a Standalone Line</td>
<td>68</td>
</tr>
<tr>
<td>Add a Table</td>
<td>71</td>
</tr>
<tr>
<td>Add a Widget</td>
<td>65</td>
</tr>
<tr>
<td>Add A Work Log Item to a Ticket (Agent or Admin Users)</td>
<td>225</td>
</tr>
<tr>
<td>Add an Anchored Line</td>
<td>67</td>
</tr>
<tr>
<td>Add an Attachment to a Ticket</td>
<td>225</td>
</tr>
<tr>
<td>Add an Image</td>
<td>70</td>
</tr>
<tr>
<td>Add Column Panes</td>
<td>205</td>
</tr>
<tr>
<td>Add Links to a Table</td>
<td>95</td>
</tr>
<tr>
<td>Add Portlets</td>
<td>20</td>
</tr>
<tr>
<td>Add Systems to a Maintenance Schedule</td>
<td>492</td>
</tr>
<tr>
<td>Add Text</td>
<td>74</td>
</tr>
<tr>
<td>Admin Console Guide</td>
<td>33</td>
</tr>
<tr>
<td>Administering Report Scheduler</td>
<td>277</td>
</tr>
<tr>
<td>Administration</td>
<td>503</td>
</tr>
<tr>
<td>Adobe Flash Player Settings Dialog</td>
<td>46</td>
</tr>
<tr>
<td>Advanced Search</td>
<td>408</td>
</tr>
<tr>
<td>Alarm Components</td>
<td>158</td>
</tr>
<tr>
<td>Alarm Console</td>
<td>35</td>
</tr>
<tr>
<td>Alarm Console Loses Filters with Internet Explorer</td>
<td>56</td>
</tr>
<tr>
<td>Alarm Console Table</td>
<td>40</td>
</tr>
<tr>
<td>Alarm Console Table Popup Menu</td>
<td>43</td>
</tr>
<tr>
<td>Alarm Console Toolbars</td>
<td>35</td>
</tr>
<tr>
<td>Alarm Data Format Pane</td>
<td>211</td>
</tr>
<tr>
<td>Alarm Filter Bar</td>
<td>37</td>
</tr>
<tr>
<td>Alarm Filter Dialog</td>
<td>46</td>
</tr>
<tr>
<td>Alarm Filter Expression Dialog</td>
<td>46</td>
</tr>
<tr>
<td>Alarm History Dialog</td>
<td>47</td>
</tr>
<tr>
<td>Alarm List</td>
<td>471</td>
</tr>
<tr>
<td>Alarm Notification Tab in SLA Form</td>
<td>317, 325</td>
</tr>
<tr>
<td>Alarm Summary Chart</td>
<td>471</td>
</tr>
<tr>
<td>Alarm Table</td>
<td>476</td>
</tr>
<tr>
<td>Align Widgets</td>
<td>99</td>
</tr>
<tr>
<td>Apply a Monitor to a System</td>
<td>466</td>
</tr>
<tr>
<td>Applying Monitors to Groups</td>
<td>461</td>
</tr>
<tr>
<td>Applying Monitors to Systems</td>
<td>466</td>
</tr>
<tr>
<td>Assign a Data Source</td>
<td>90</td>
</tr>
<tr>
<td>Assign a Host Variable</td>
<td>146</td>
</tr>
<tr>
<td>Assign a Monitoring Template</td>
<td>463</td>
</tr>
<tr>
<td>Assign a QoS Variable</td>
<td>145</td>
</tr>
<tr>
<td>Assign a Robot Variable</td>
<td>145</td>
</tr>
<tr>
<td>Assign a Text Variable</td>
<td>146</td>
</tr>
<tr>
<td>Assign To Dialog</td>
<td>47</td>
</tr>
<tr>
<td>Assigning Report Templates to Groups</td>
<td>490</td>
</tr>
<tr>
<td>Assigning Variables</td>
<td>144</td>
</tr>
<tr>
<td>Asynchronous data type</td>
<td>390</td>
</tr>
<tr>
<td>Attach Notes Dialog</td>
<td>47</td>
</tr>
<tr>
<td>Automatic (Interval) data type</td>
<td>390</td>
</tr>
<tr>
<td>Automatic Groups</td>
<td>447</td>
</tr>
<tr>
<td>Automatic Replacement of Fields in PRD Templates</td>
<td>241</td>
</tr>
<tr>
<td>Automatic Robot Deployment</td>
<td>429</td>
</tr>
<tr>
<td>Automatic Weight Distribution</td>
<td>333</td>
</tr>
<tr>
<td>Automatically Create Tickets from Alarms</td>
<td>288</td>
</tr>
<tr>
<td>Background Alpha</td>
<td>175</td>
</tr>
<tr>
<td>Background Color</td>
<td>174</td>
</tr>
<tr>
<td>Background ImageURL</td>
<td>176</td>
</tr>
<tr>
<td>Badge View</td>
<td>403</td>
</tr>
<tr>
<td>Baseline Value Definition</td>
<td>355</td>
</tr>
<tr>
<td>Basic Search</td>
<td>407</td>
</tr>
<tr>
<td>Best Practices for Creating Ranges</td>
<td>418</td>
</tr>
</tbody>
</table>
C

CA APM Cloud Monitor • 57
Calculation Examples • 306
Calculation method • 390
Calculation Profiles • 388
Calculation Terms and Conditions • 302
Calculation Terms and Conditions for the QoS • 303
Calculation Terms and Conditions for the SLA • 305
Calculation Terms and Conditions for the SLO • 304
Calculations Tab in SLA Form • 319
Change Layout Style Selector • 258
Change Names of Custom Fields • 475
Change Password • 31
Change the Appearance of a Widget • 96
Change the Type of Widget • 100
Changing What You See • 255
Charts Pane • 236
Clone a Widget • 98
Close Browser When Logging Out • 513
Cloud Monitor Server Location Configuration • 58
CMDB Integration • 289
Command Bar • 36
Compliance Percentage • 301, 391
Compliance period • 390
Configure Email Server Settings • 277
Configuring a Widget to Use External DB Data • 142
Configuring Access to the CA APM Cloud Monitor Portlet • 59
Configuring the Table Widget • 140
Contact CA • 3
Converting Data Units • 130
Copy a Job • 275
Copy a Monitoring Template • 464
Create a Dashboard
  Overview • 62
Create a Dashboard Data Source • 84
Create a Filter • 449
Create a Group • 446
Create a Monitoring Template • 462
Create a New Service Level Objective (SLO) • 321
Create a Page • 20
Create a Probe Data Source • 86
Create a QOS Data Source • 85
Create a Range • 418
Create a Variable • 144
Create an Alarm Data Source • 83
Create an SLA or SLO Data Source • 85
Create an SQL Data Source • 88
Create and Assign a Data Source • 82
Create Authentication Profiles • 413
Create Dashboard • 113
Create Groups Automatically • 448
Create NM Users • 290
Create or Edit a Job • 273
Create or Edit a Maintenance Schedule • 492
Create Service Desk Users • 290
Create Tickets from Alarms • 288
Creating a Calculation Profile • 336
Creating a Compliance Period • 334
Creating a Geo View • 411
Creating a QoS Monitoring Profile • 355
Creating an Exclude Period • 342
Creating an Exclude Period, using the Add Exclude Period Wizard • 343
Creating an Operating Period • 335
Creating an SLA • 313
Creating an SLA Using the SLA Wizard • 348
CSV File is Incorrect in Chinese or Japanese • 519
Custom Dashboards (Legacy) • 107
Custom Logo Missing • 510
Customize Data Sources for Systems • 185
Customizing the Dynamic View Templates • 188

D

Dashboard • 61
Dashboard API on Windows Does Not Work on Linux • 182
Dashboard Components Pane • 158
Dashboard Designer • 167
Dashboard Designer (Legacy) • 111
Dashboard Objects Properties • 163
Dashboard Pane • 107, 168
Data Collection and Compliance Calculation • 301
Data Management • 364
Data Sources for Dynamic Views • 185
Data Types • 392
Default Fonts for Reports • 279
Define an Image Map for an Image Widget • 93
Define Ranges • 417
Define Reports with a URL • 246
Define the XML File • 431
Defining a Quality of Service Object • 329
Defining QoS Constraints • 329
Delete a Dashboard • 103
Delete a Data Source • 91
Delete a Group • 455
Delete a Job • 275
Delete a Monitor from a System • 467
Delete a Variable • 147
Delete a Widget • 100
Delete Text from Custom Fields • 475
Deploy Robots Automatically • 430
Deploy Robots to Systems Listed in a File • 431
Deploying Widgets • 117
Design a List • 198
Designing a New Dashboard • 113
Detach Notes dialog • 48
Details Dialog • 48
Device Performance Charts Reference • 483
Dialogs • 45
Disable Public Reports • 215, 245
Discovery • 412
Display Metrics for Custom Probes • 486
Distributing Importance (Weight) • 333
Do Not Edit QoS Properties in SLM Portlet • 513
Drilling Down in List Viewer • 217
Duplicate Data Series are generated • 193
Dynamic View is not Displayed • 193
Dynamic Views • 183
Dynamic Views May Display Redundant Tree Nodes • 192
Edit a Dynamic View Template • 192
Edit a Monitor for a System • 468
Edit a Variable • 146
Edit ACLs Dialog • 27
Edit Alarm Filter Dialog • 28
Enabling the Alarm Console • 55
Error Console • 21
Errors • 498
Example • 147, 450
Example – Creating a SLA Based on the Selected SLA • 354
Example – Creating an SLA Based on a Service • 348
Example - Defining a Simple Service Level Agreement • 345
Example 1
One QoS and One SLO • 307
Example 2
Two QoSs and One SLO • 308
Example 3
Two QoSs and One SLO, Using Calculation Method AND or OR • 308
Example 4
Two SLOs, Each with Three QoS • 309
Excluded Periods • 320
Excluding Quality of Service Data • 341
Export a Dashboard • 104
Export Alarms to CSV • 476
Export Search Results to CSV • 409
Exporting QoS Data Using Export QoS Data Wizard • 364
Exporting the QoS Data Series • 363
Filter Alarms • 479
Filter Alarms by Date • 480
Filter Menus and Buttons • 453
Filters • 449
Filters Tab • 234
Find Dialog • 48
Flow Analysis • 195
Garbled Text in Console Installer • 512
Gauge Data Format Pane • 208
General Bar • 35
General Properties • 178
Generating a Graph • 362
Generating a Report as a PDF • 297
Geo View • 410
Get Link URL Button • 264
Getting Started with Service-Oriented Configuration • 443
Getting Started with UMP • 17
Group Not Found • 499
Guest Dashboards Not Displayed • 109
History Bar • 38
How to Navigate the Help Topics • 22
How to Print Multiple Help Topics • 23
How to Use Parameters • 101
Hub • 392
I
I Cannot Access a Wiki Page I Created • 511
I Cannot Access the Control Panel • 518
I Cannot Save My Information • 420
I See Messages About Exclude Scopes (Ranges) • 420
Import a Dashboard • 104
### P

- Page Titles May Contain Garbled Text • 512
- Paging • 403
- Panel • 160
- PDF Preferences Dialog • 214, 244
- PDF Report Depth • 298
- Perform Ad Hoc Maintenance • 493
- Performance Reports Designer • 231
- Performance Reports May Convert Incorrectly to PDF • 252
- Pop Out Graphs • 409
- Popup Menus • 42
- Portlet Not Resizing in Chrome • 515
- Position a Widget • 91
- Precedence • 457
- Preferences Dialog • 51, 219, 246, 297, 382
- Prepare the Canvas • 63
- Preparing the Canvas • 134
- Preview a Job • 274
- Priority • 458
- Probe Updates • 504
- Probes • 393
- Properties Pane • 174
- Publish a Dashboard • 103

### Q

- QoS • 394
- QoS constraint • 394
- QoS Data Tables • 374
- QoS Definition Tab • 369
- QoS Monitor Properties • 357
- QoS Monitors List • 356
- QoS object • 394
- QoS Objects • 296
- QoS Selection Tabs • 233
- Quality of Service • 389
- Quality of Service Reports • 269
- Query Dialog • 52

### R

- Radar View Button • 258
- Radius Slider • 261
- Reapply a Monitoring Template • 465
- Relationship Selector • 261
- Relationship Services Not Supported on Solaris • 511
- Relationship Viewer • 253
- Relationship Viewer Control Bar • 256

Relationship Viewer Dependent upon Root Cause Analysis and Topology Manager • 513
Report Scheduler • 273
  - Removing an Account User's Access • 282
Report Scheduler Does Not Display Estimated Values • 283
Report Scheduler Jobs Table • 276
Report Scheduler Shifts Time Axis in PDF Format • 283
Reports • 269
  - Reports Overview • 269
  - Reports with Integrated Data • 289
Requests List • 224
Resize a Widget • 96
Right-Clicking Objects on the Canvas • 179
Robot • 393
Run a Job • 275
Run File-based Import • 421
Run Query Dialog • 52
Running the Discovery Wizard • 412

### S

- Save a Dashboard • 102
- Save Dialog • 214, 244
- Scaling Dashboards • 177
- Schedule Discovery • 419
- Search and Clear Search Controls • 261
- Search for Computer Systems • 407
- Select a Layout • 20
- Selecting a Data Source--DB • 139
- Selecting a Data Source--NIS • 136
- Selecting a Data Source--Probe • 137
- Selecting a Data Source--QoS • 135
- Selecting a Data Source--SLA • 139
- Selecting a Data Source--Variables • 137
- Selecting the Data Source • 134
- Sending SQL Queries to the Database • 377
- Servers Group • 455
- Service Desk • 287
- Service Level Agreement (SLA) • 393
- Service Level Agreement Form • 314
- Service Level Agreement Reports • 272
- Service Level Agreements • 384
- Service Level Objective Form • 322
- Set a Background Image • 64
- Set a Custom Font for Reports • 281
- Set a Monitor Override • 468
- Set a Single Alarm to Invisible • 478
Set a URL for a Widget • 92
Set Custom Alarm Fields • 474
Set Custom Fields Dialog • 53
Set Default View Dialog • 53
Set Multiple Alarms to Invisible • 478
Set Permissions for UMP Portlets • 504
Set Properties for a Widget • 91
Set Sounds for a Widget • 94
Set the Background Properties • 64
Set the Color • 97
Set the Grid Functions • 65
Set the Monitoring Template State for a Group • 460
Set the Opacity • 96
Set the State for a System • 460
Set the Width of a Line • 98
Set Up Service-Oriented Configuration • 443
Set Where Reports are Stored on a Server • 277
Setting Up a Probe to Deliver Quality of Service Data • 312
Single Sign On • 503
Single Time Zone Required • 509
Size Limits for SQL Queries • 378
Size of Dashboard Panels • 513
Size the Canvas • 63
SLA Calculation Tables • 375
SLA Configuration Tables • 375
SLA Reports • 293
SLM • 299, 393
SLM Database Structure • 374
SLM Interface Reference • 379
SLM May Allow Invalid Logical Expressions • 394
SLM Overview • 300
SLM Portlet Displays Incorrect Data Set • 395
SLM Terms and Definitions • 390
SLO • 393
SLOs • 296
Slow Performance on Internet Explorer 7/8 • 517
SNMP • 415
SOC Probes Configuration Reference • 444
Some Scheduled Jobs are Not Running • 284
Sort Alarms • 481
Sorting • 405
Source Data Format Pane • 213
Spark Data Format Pane • 207
Specify a Group AND a System • 497
Specify a Group OR a System • 496
Specify a Path • 497
State • 459
Status Data Format Pane • 212
Step 1 – Define an Operating Period • 345
Step 2 – Create a new SLA and Define the Service Level Compliance Requirement • 346
Step 3 – Define the Individual Service Level Objectives • 347
Step 4 – Excluding Periods • 348
Supported Variables • 190
System Not Found • 498

T

Template Names • 188
The Active Objects Tab • 365
The Application Window • 157
The Calculation Settings Tab • 328
The Dashboards Templates • 165
The Excluded Periods Tab • 327
The Menu Bar • 380
The Navigation Pane • 383
The Portal • 17
The QoS Probes Tab • 370
The Quality of Service Tab • 323
The SLM Portlet Does Not Record a QoS Object • 395
The Unavailability Report • 272
Time Slice Column Does Not Display Any Data • 216
Time Slice Column Is Missing Data • 216
Time Slice Data Format Pane • 209
Toolbar • 242
Toolbar Popup Menu • 42
Trace Log Dialog • 54
Transaction History Dialog • 54
Tree View • 403
Troubleshooting • 509
Troubleshooting Alarm Console • 55
Troubleshooting Dashboard Designer • 181
Troubleshooting Dynamic Views • 192
Troubleshooting List Designer • 215
Troubleshooting Nimsoft Remote Admin • 228
Troubleshooting PRD • 251
Troubleshooting Report Scheduler • 282
Troubleshooting SLM • 394
Troubleshooting the Discovery Wizard • 420
Turn Off Compatibility View with IE 8 or 9 • 514
Types of Users • 25

U

UMP Views • 29
Unassign a Data Source • 90
Understanding Performance Data • 231
Understanding Precedence, Priority, and State • 457
Understanding SLA Reports • 295
Understanding the Dynamic Views Tree • 184
Undoing Operations • 166
Unexpected Character Appears in Reports • 284
Unified Dashboards Guide • 501
Unified Reporter • 397
Unified Service Manager • 399
Unified Service Manager Displays Non-localized Text • 510
Url Attribute • 132
User Names • 515
User Tags • 200, 235, 452
User Types • 223
Users in Service Desk • 290
Using ArrowLine and EllipseLine • 121
Using Line Map • 125
Using SLM • 311
Using the Contents Feature • 22
Using the Index Feature • 23
Using the Relationship Viewer • 255
Using the Search Feature • 23
Using Variables for Widgets • 143
USM Alarm and Device Icons • 405
USM Templates Do Not Deploy • 229

V

Variables Pane • 181
View a Dynamic View • 183
View a List of Widgets on the Canvas • 100
View Alarms • 470
View Discovered Systems • 428
View Group Assignments • 455
View Job History • 276
View Metrics • 485
View Metrics for Virtual Machines • 485
View Modes • 63
View Monitor Status • 469
View System Information in USM • 187
View System Properties • 481
View the Job Log File • 276
View the Status of Robot Deployment • 442
Viewing Actual Database Usage • 364
Viewing Object Details • 264
Viewing Raw Performance Data • 232
Viewing Reports • 491
Viewing SLA History • 294
Viewing SLA Reports • 295
Viewing System Information • 481
Viewing the Current Sample Values • 332
Viewing the Data as a Performance Report • 188
Viewing the Quality of Service (QoS) Data • 361
Viewing Tickets • 224
Views • 402

W

Watching SLA Calculations • 372
Welcome • 17
Welcome to Unified Management Portal Help • 22
Which Item is Displayed? • 498
WMI • 417
Working with Alarms • 470
Working with Groups • 445
Working with Invisible Alarms • 477
Working with Monitors • 456
Working with Report Scheduler • 273
Working with Reports • 490

X

XML File Schema • 422

Z

Zoom Slider • 260