



Silect MP Author for Microsoft
System Center Operations Manager

Silect Software's MP Author/MP Author Pro User Guide

Oct 15, 2018

Disclaimer

The information in this document is furnished for informational use only and is subject to change without notice. Silect Software, Inc. assumes no responsibility or liability for any errors or inaccuracies that may appear in this document.

The information contained herein is the property of Silect Software, Inc. and is strictly confidential. Except as expressly authorized in writing by Silect Software, Inc., the holder of this document shall keep all information contained herein confidential. Except as expressly authorized in writing by Silect Software, Inc., the holder is granted no rights to use, reproduce, or otherwise disclose or disseminate the information contained herein.

Copyright © 2018
Silect Software, Inc.
6333 Rideau Valley Dr. N.

Manotick, Ontario K4M 1B3
Canada

www.silect.com

Contents

Contents	2
Preface	4
About Silect Software's MP Author User Guide	4
Audience	4
Document conventions	4
Command syntax	4
Notes, tips and cautions	5
Chapter 1	6
About Silect MP Author/MP Author Pro	6
Hardware and software requirements	6
Differences Between MP Author & MP Author Pro	6
Chapter 2	7
Installing and configuring Silect MP Author	7
Prerequisites	7
Installing and Uninstalling Silect MP Author	7
Configuring Silect MP Author	7
Configuring Operations Manager to Monitor SNMP Devices	8
Chapter 3	9
Working with Silect MP Author	9
Starting Silect MP Author	9
To Open the Silect MP Author Console	9
To Troubleshoot Silect MP Author Startup Problems	9
Creating New Management Packs	9
To Start the Management Pack Creation Wizard	10
Empty Management Pack	11
Single Server Management Pack	11
Distributed Application Management Pack	14
Customizing Existing Management Packs	15
Adding and Editing Folders	16
Adding and Editing Classes and Discoveries	17
Adding and Editing Rules	24
Adding and Editing Monitors	28
Adding and Editing Views	36
Adding and Editing Relationships	39
Adding and Editing Secure References	40
Adding and Editing Tasks	41
Adding and Editing Diagnostics and Recoveries	42
Authoring with Fragments	44
Creating Fragments	45
Editing Fragments	47

Managing Fragments	48
Management Pack Options	49
Chapter 4	55
Getting Technical Support	55
Self-Service Support Portal	55
Electronic Support	55
Premium Support	55
Other Resources	55
Appendix A: Fragments	56
Built-In Parameters	56
Browsable Parameters	56
Special Parameter Rules	57
Sanitizing	57
Required References and Aliases	58

Preface

About Silect Software's MP Author User Guide

The Silect Software's MP Author User Guide describes how to install, configure, work with and troubleshoot MP Author for use with Microsoft System Center Operations Manager. It contains the following chapters:

- About Silect MP Author/MP Author Pro
- Installing and configuring Silect MP Author
- Working with Silect MP Author
- Getting Technical Support

Audience

This document is intended for the following users:

- Network planners
- Installers
- Administrators
- Operators
- Maintenance personnel

Document conventions

This document uses **bold** to identify the following:

- File and directory names, as follows:
For details, refer to the **readme.txt** file.
- User interface names, titles, buttons, and tabs, as follows:
To display the **Open** dialog, choose **File, Open...**
- Command names, as follows:
To display a list of available diagnostics, use the **show list** command.

Command syntax

Table 1: Command syntax rules describes the command syntax rules used in this document.

Table 1: Command syntax rules

Syntax format	Description
Bold	Commands and keywords . Enter command line arguments that are in bold exactly as written.
<variable>	Variables. You must substitute a value for command line arguments that are enclosed in angle brackets.
[optional keyword]	Optional keywords or <variables>. Enter command line

[<optional variable>]	arguments that are enclosed in square brackets, if desired.
{ a b c } {<a> <c>}	Required alternate keywords or <variables>. You must choose one command line argument enclosed within the braces. Choices are separated by vertical (OR) bars.
[a b c] [<a> <c>]	Optional alternate keywords or <variables>. Choose one command line argument enclosed within the braces, if desired. Choices are separated by vertical (OR) bars.

Notes, tips and cautions

Notes and cautions are used to alert you to important information or risks.

Notes

Notes use the following convention:

Note	Take note. Contains important and helpful information.
-------------	--

Tips

Tips use the following convention:

Tip	Review and consider. An alternative or faster method is being suggested.
------------	--

Cautions

Cautions use the following convention:

CAUTION!	Be careful. In this situation, you might do something that could result in a service interruption, or the loss of data.
-----------------	---

Chapter 1

About Silect MP Author/MP Author Pro

Silect Software's MP Author is a management pack (MP) design, development and customization software program for Microsoft System Center Operations Manager (SCOM) 2012, 2016, 1801 or 1807. It allows users to create completely new MPs or customize existing MPs—whether they are provided by Microsoft or by other third-party vendors.

This chapter provides an overview of the functionality of Silect MP Author. It contains the following sections:

- [Hardware and software requirements](#)
- [Differences Between MP Author & MP Author Pro](#)

Hardware and software requirements

Please refer to the MP Author Installation Guide for hardware and software requirements.

Differences Between MP Author & MP Author Pro

There is a single installation used for both MP Author and MP Author Pro – the software for the two versions are identical. The only difference between the two is that MP Author Pro has a valid license file, which if it exists, is used to turn on the professional level authoring features, not available in the free version. If you purchase MP Author Pro this license file will be provided to you by Silect.

When this document describes features in MP Author, these features will also be included in MP Author Pro. MP Author Pro contains all the features in MP Author, along with additional advanced authoring capabilities not available in the free MP Author product. MP Studio also provides the same level of professional authoring provided by MP Author Pro.

Chapter 2

Installing and configuring Silect MP Author

This chapter describes the requirements and procedures for installing, configuring and uninstalling Silect MP Author. It contains the following sections:

- Prerequisites.
- Installing Silect MP Author.
- Configuring Silect MP Author.
- Uninstalling Silect MP Author.
- Configuring Operations Manager to Monitor SNMP Devices.

Prerequisites

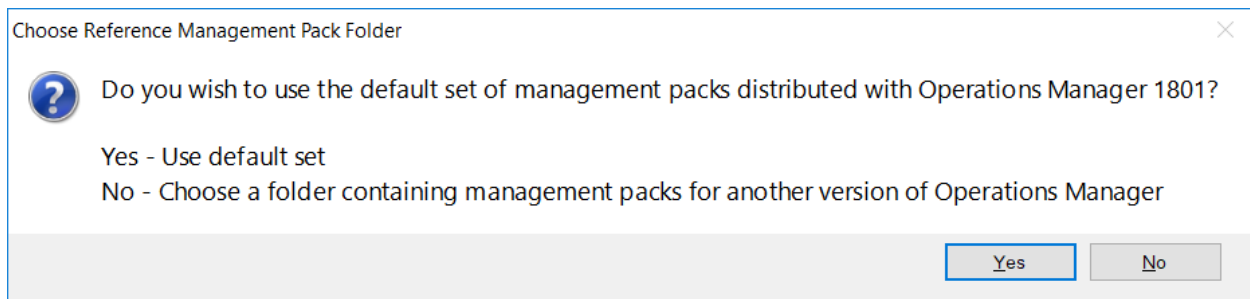
Please refer to the MP Author Installation Guide for prerequisites.

Installing and Uninstalling Silect MP Author

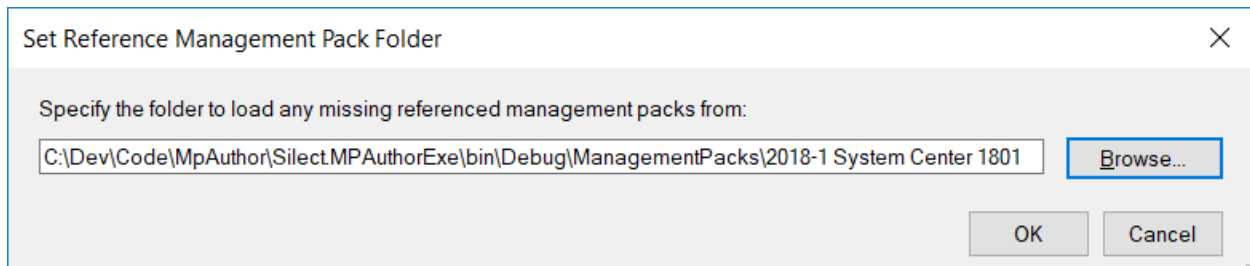
Please refer to the MP Author Installation Guide for installation and removal instructions.

Configuring Silect MP Author

When Silect MP Author is started, it will launch and show the **Welcome to MP Author** page. The following dialog will be shown asking which set of management packs to use:



If you wish to use the management packs distributed with the Operations Manager 1801 release (or 1807 – they use the same MPs), click **Yes**. If you click **No**, the following dialog will be shown:



Silect MP Author requires access to reference management packs to create or edit other management packs. Click the **Browse** button to select the folder where the System Center Operations Manager reference management packs reside and click **OK**. Management packs for the original Operations Manager 2012 release, as well as for SP1, R2, 2016 and 1801 are provided in subfolders. SCOM 1807 uses the same MPs as SCOM 1801.

Tip Having the reference management packs reside locally allows for the use of Silect MP Author at any time.

Note The reference management pack folder can be changed at any time using the **Tools, Set Reference Management Pack Folder** option.

Configuring Operations Manager to Monitor SNMP Devices

Just as Operations Manager needs to be configured to monitor specific computers and servers, it must also be configured to monitor specific SNMP devices. This must be done using the SCOM console as follows:

- Step 1 Start the System Center Operations Manager Console.
- Step 2 Launch the discovery wizard and select Network Devices.
- Step 3 Specify a name for your network discovery process, select a management server to run your discovery process, create a management resource pool specifically for network discovery and select it.
- Step 4 Next choose either explicit discovery or recursive. If you wish to just work with just one single device, then explicit should be fine. If you want to work with a group of network devices, then you should select recursive (and plan on working from the root SNMP device on your network).
- Step 5 Specify a **Run As** account for your network device credentials. This will be used to specify the credentials used for your discovery process. You can specify multiple **Run As** account if you have multiple network devices.
- Step 6 Type Add and then specify the IP address of your network device, along with any other settings required if the default ones are not appropriate.
- Step 7 You next need to specify a schedule at which to run your network device discovery process. Typically, it is run once a day.
- Step 8 The discovery should be run immediately once you have completed defining it, and if everything was defined okay, you should see your network devices showing up in the Operations Manager console shortly.

If you have any difficulty getting Operations Manager to recognize your SNMP devices and management packs, we recommend that you read some of the following articles:

<https://blogs.technet.microsoft.com/kevinholman/2011/07/20/opsmgr-2012-discovering-a-network-device/>

This article talks specifically about configuring Operations Manager to get it working with SNMP traps:

<https://blogs.technet.microsoft.com/kevinholman/2015/02/03/snmp-trap-monitoring-with-scom-2012-r2/>

Chapter 3

Working with Silect MP Author

This chapter describes how to work with Silect MP Author. It contains the following sections:

- Starting Silect MP Author.
- Creating new management packs.
- Customizing existing management packs.
- Management Pack Options.

Starting Silect MP Author

Once Silect MP Author has been installed, it will be added to the Start menu and as well a shortcut will be added to the desktop on the system where it is installed. This section describes how to start Silect MP Author and how to troubleshoot if Silect MP Author doesn't start properly.

Note Before starting the Silect MP Author console for the first time ensure the location of the reference management packs is known.

To Open the Silect MP Author Console

- Step 1 Click the **Start** menu, locate and select **Silect** and click **MP Author**.
- Step 2 The Silect MP Author console should appear and the **Welcome to MP Author** page will be displayed.

To Troubleshoot Silect MP Author Startup Problems

If there are errors or problems when starting Silect MP Author, please ensure the hardware and software prerequisites are met including the installation and operation of the System Center Operations Manager console on the system where Silect MP Author is installed. If these prerequisites have been met and you are still experiencing problems, please see the section entitled Contacting Technical Support for further assistance.

Creating New Management Packs

There are several scenarios where the creation of a new management pack (MP) is required. For example, an MP may not exist for a specific application you need to monitor with Operations Manager. Or perhaps the MP that came with a system you are managing is not meeting your needs and a new MP with more (or less) information can optimize the management of the system.


Silect MP Author supports a wizard-based MP creation capability. You will be guided through creating the key elements of an MP and when the wizard completes you can further modify the MP to meet your requirements. In virtually all cases the Management Pack Creation Wizard allows you to browse the

application or system you are building the MP for which provides an easier, more accurate means of defining the necessary MP elements.

Tip Before starting the **Management Pack Creation Wizard**, ensure you have network access to the system or application you are creating an MP for. Credentials used for browsing can be managed using the **Tools, Manage Server Credentials** menu command.

Note It is recommended to have a separate file folder structure for each MP that you plan on creating. Creating separate MPs for each application or device is recommended, rather than having one MP which monitors several unrelated applications and/or devices.

To Start the Management Pack Creation Wizard

- Step 1 From the Silect MP Author console click the **New** toolbar icon () or go to **File, New**.
- Step 2 The New **Management Pack Creation Wizard, Introduction** screen will be displayed. Click **Next**.
- Step 3 Input manifest information for the management pack including name, display name, version and a description. Default values are provided for version and description, but you may want to change them. Click **Next**.
- Step 4 Specify or use the **Browse** button select an existing folder to be used as the management pack working folder. Click **Next**.

Note It is recommended to have a separate file folder structure for each MP that you plan on creating.

- Step 5 The **Add References** wizard screen will be displayed, and a default set of referenced MPs will be displayed. If additional referenced MPs need to be added, click the **Add Optional References** button and choose additional reference MPs. Click **Next**.

Note that MP Author Pro users will be provided with additional selection options on the Add References dialog to allow them to select which set of management packs they will be referencing. There are special sets that must be selected when authoring SNMP management packs.

To select different sets of referenced management packs, click on one of the items below.

- Windows (SCOM 1801 or 1807)
- Windows (SCOM 2016)
- Windows (SCOM 2012 R2)
- Windows (SCOM 2012 SP1)
- Windows (SCOM 2012)
- SNMP (SCOM 1801 or 1807)
- SNMP (SCOM 2016)
- SNMP (SCOM 2012 R2)
- SNMP (SCOM 2012 SP1)
- SNMP (SCOM 2012)

At this point you will be asked to select an MP template to further assist the creation of a new management pack. You are given three choices:

- **Empty Management Pack:** Select this option if you are a more advanced user and would like to work with an empty MP with minimal details required by the wizard. The Empty Management Pack template requires the user to manually configure all elements of the MP.
- **Single Server Application:** Select this option to create a new MP that targets a single component (application or Windows Service etc.).
- **Distributed Application:** Select this option to create a new MP that targets a distributed application. Distributed applications typically have a back-end database, service middle tier and a client front-end.

Follow the appropriate section below that corresponds to the type of MP you are creating.

Empty Management Pack

This option is for an advanced user who wants to manually create all elements of an MP. To create a new empty MP, follow these steps:

- Step 1 From the **Select the MP Template to Use** wizard screen select the **Empty Management Pack** option. Click **Next**.
- Step 2 At the **Create the Management Pack** wizard screen click **Finish**.

At this point a new MP will be created and you will be returned to the Select MP Author designer window to manually make changes to the MP.

Single Server Management Pack

This option is for an advanced user who wants to create an MP for a simple application or system consisting of a single component. To create a new single server MP, follow these steps:

- Step 1 From the **Select the MP Template to Use** wizard screen select the **Single Server Application** option. Click **Next**.

Note You will next be guided through the creation of classes with discoveries and monitors for your single server MP.

- Step 2 At the **Define the Role (Target) the MP will monitor** wizard screen, select the appropriate setting used to discover the application. Choices are **Registry Setting**, **Script Setting** or **WMI Setting**. There are other types of discoveries which can be added, once the **New Management Pack Wizard** is complete.

Follow the appropriate section below that corresponds to the type of role being added to the MP.

Creating a New Windows Registry Target

A registry target is used to identify the existence of an application or system using a registry entry or value. To create a new registry target:

- Step 1 At the **Define the Role (Target) the MP will monitor** wizard screen click **Use Registry Setting**.
- Step 2 Follow the instructions, starting at Step 2 from [Creating a New Windows Registry Target](#) below.

When you have completed adding a new registry target you will be returned to the Management Pack Creation Wizard. Click **Next** to create monitors to be added to the MP. See the sections below on adding monitors to the MP.

Creating a New Windows Script Target

A script target is used to identify the existence of an application or system using the results of running a script. To create a new script target:

- Step 1 At the **Define the Role (Target) the MP will monitor** wizard screen click **Use Script Setting**.
- Step 2 Follow the instructions, starting at Step 2 from [Creating a New Windows Script Target](#) below.

When you have completed adding a new script target you will be returned to the Management Pack Creation Wizard. Click **Next** to create monitors to be added to the MP. See the sections below on adding monitors to the MP.

Creating a New Windows WMI Target

A WMI target is used to identify the existence of an application or system using information from the WMI database. To create a new WMI target:

- Step 1 At the **Define the Role (Target) the MP will monitor** wizard screen click **Use WMI Setting**.
- Step 2 Follow the instructions, starting at Step 2 from [Creating a New Windows WMI Target](#) below.

When you have completed adding a new WMI target you will be returned to the Management Pack Creation Wizard. Click **Next** to create monitors to be added to the MP. See the sections below on adding monitors to the MP.

Monitors

At the **Define the Monitors for this MP** wizard screen click **New** and select the appropriate type of monitor to add to the MP. Choices are **Database Monitor**, **Event Monitor**, **Performance Monitor**, **Script Monitor**, **Service Monitor** and **Web Site Monitor**. There are other types of monitors which can be added, once the **New Management Pack Wizard** is complete.

Follow the appropriate section below that corresponds to the type of monitors being added to the MP.

Creating a New Windows Database Monitor

A database monitor is used to determine the health state of a database component either on its own or part of a larger application or system. To create a new database monitor:

- Step 1 At the **Define the Monitors for this MP** wizard screen click **New**, then click **New Database Monitor**.
- Step 2 Follow the instructions, starting at Step 2 from [Creating a New Windows Database Monitor](#) below.

When you have completed adding a new database monitor you will be returned to the **Define the Monitors for this MP** wizard screen. You can choose to create another monitor, or you can complete the creation of the MP by clicking **Next**.

Creating a New Windows Event Monitor

An event monitor is used to determine the health state of an application or system based on Event Log entries. To create a new event monitor:

- Step 1 At the **Define the Monitors for this MP** wizard screen click **New**, then click **New Event Monitor**.
- Step 2 Follow the instructions, starting at Step 2 from [Creating a New Windows Event Monitor](#) below.

When you have completed adding a new event monitor you will be returned to the **Define the Monitors for this MP** wizard screen. You can choose to create another monitor, or you can complete the creation of the MP by clicking **Next**.

Creating a New Windows Performance Monitor

A performance monitor is used to determine the health state of an application or system based on Windows performance information. To create a new performance monitor:

- Step 1 At the **Define the Monitors for this MP** wizard screen click **New**, then click **New Performance Monitor**.
- Step 2 Follow the instructions, starting at Step 2 from [Creating a New Windows Performance Monitor](#) below.

When you have completed adding a new performance monitor you will be returned to the **Define the Monitors for this MP** wizard screen. You can choose to create another monitor, or you can complete the creation of the MP by clicking **Next**.

Creating a New Windows Script Monitor

A script monitor is used to determine the health state of an application or system based on the results of running a script. To create a new script monitor:

- Step 1 At the **Define the Monitors for this MP** wizard screen click **New**, then click **New Script Monitor**.
- Step 2 Follow the instructions, starting at Step 2 from [Creating a New Windows Script Monitor](#) below.

When you have completed adding a new script monitor you will be returned to the **Define the Monitors for this MP** wizard screen. You can choose to create another monitor, or you can complete the creation of the MP by clicking **Next**.

Creating a New Windows Service Monitor

A service monitor is used to determine the health state of an application or system based on the state of a Windows service (running or not running). To create a new service monitor:

- Step 1 At the **Define the Monitors for this MP** wizard screen click **New**, then click **New Service Monitor**.

- Step 2 Follow the instructions, starting at Step 2 from [Creating a New Windows Service Monitor](#) below.

When you have completed adding a new service monitor you will be returned to the **Define the Monitors for this MP** wizard screen. You can choose to create another monitor, or you can complete the creation of the MP by clicking **Next**.

Creating a New Windows Web Site Monitor

A web site monitor is used to determine the health state of a web service component either on its own or part of a larger application or system. To create a new web site monitor:

- Step 1 At the **Define the Monitors for this MP** wizard screen click **New**, then click **New Web Site Monitor**.
- Step 2 Follow the instructions, starting at Step 2 from [Creating a New Windows Web Site Monitor](#) below.

When you have completed adding a new web site monitor you will be returned to the **Define the Monitors for this MP** wizard screen. You can choose to create another monitor, or you can complete the creation of the MP by clicking **Next**.

Rollup Algorithm

On the **Define the Rollup Algorithm** wizard screen, select the algorithm to use. **Worst Of** is the most common. Click **Next**.

Review the settings and click **Finish** to complete the creation of the new management pack.

Distributed Application Management Pack

This option is for an advanced user who wants to create an MP for an application or system consisting of a front-end or client component, a middle-tier component and a back-end or database component. The Distributed Application wizard will guide you through the creation of targets (classes/discoveries) used to discover these components and monitors used to determine the health of these components.

The wizard sequence is as follows:

Create Targets

- Step 1 Create targets for Front-End (at least one target must be defined)
- a. Create Registry Targets;
 - b. Create Script Targets;
 - c. Create WMI Targets;
- Step 2 Create targets for Middle-Tier as in Step 1 (this step is optional)
- Step 3 Create targets for Back-End as in Step 1 (at least one target must be defined)

Create Monitors:

- Step 4 Create monitors for Front-End (at least one monitor must be defined)
- a. Create Database Monitors
 - b. Create Event Monitors
 - c. Create Performance Monitors
 - d. Create Script Monitors
 - e. Create Service Monitors

f. Create Web Site Monitors

- Step 5 Create monitors for Middle-Tier as in Step 4 (this step is optional)
- Step 6 Create monitors for Back-End as in Step 4 (at least one monitor must be defined)

For complete information on creating targets, please refer to these sections:

- [Create Registry Targets](#)
- [Create Script Targets](#)
- [Create WMI Targets](#)

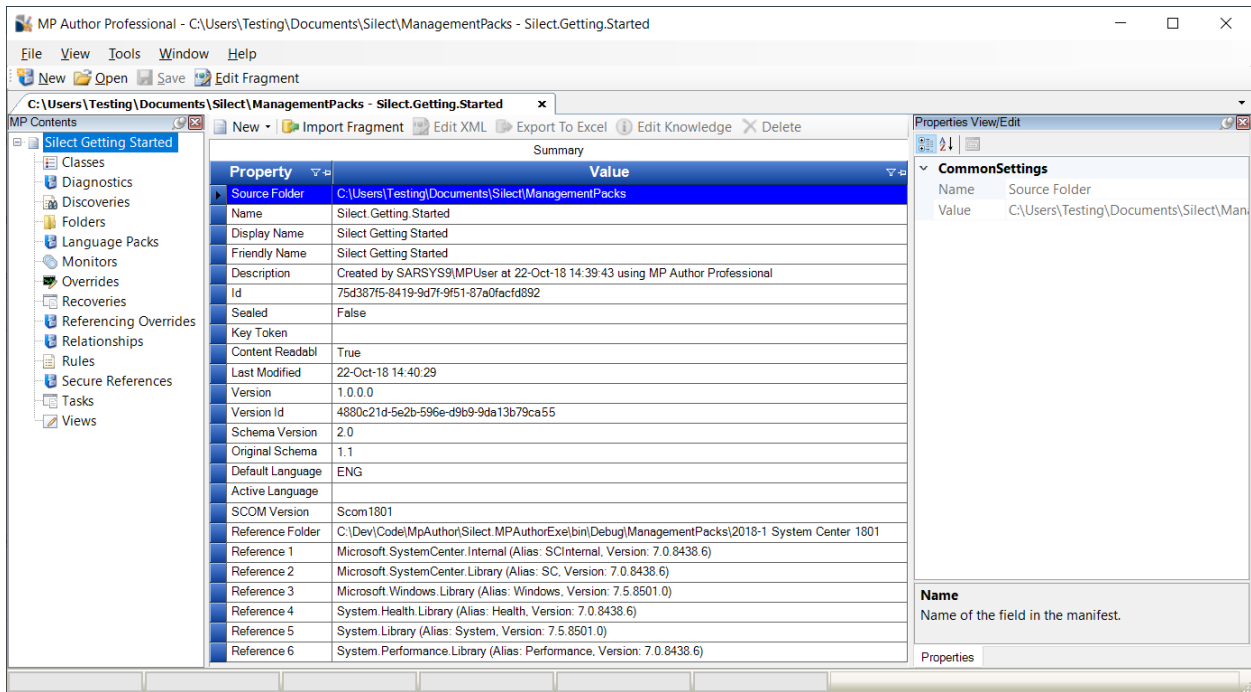
For complete information on creating monitors, please refer to these sections:

- [Create Database Monitors](#)
- [Create Event Monitors](#)
- [Create Performance Monitors](#)
- [Create Script Monitors](#)
- [Create Service Monitors](#)
- [Create Web Site Monitors](#)

Customizing Existing Management Packs

Silect MP Author provides the ability to edit existing unsealed management packs. MP objects can be added, removed and edited using Silect MP Author.

To edit an MP, from the Silect MP Author console click **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP. The MP will be opened in the designer window. Folders or containers that represent different types of MP elements (such as classes and monitors) will be displayed along the left-hand side (the left pane) of the console. Information about a selected object will be displayed in the middle pane and the properties of a selected object will be displayed in the right-hand pane. For example:



To make changes to the MP, right-click the appropriate object container and select the appropriate options. For example, for MP features such as importing into Operations Manager, right-click on the top-level node in the left-hand tree and select the appropriate option. One of the top-level options is to import a fragment, which is discussed in [Authoring With Fragments](#) below. To view options associated with an object container right click on that container.

The **Edit with Wizard...** option is available to edit any elements created with MP Author wizards. It is also available if the element appears to be compatible with the wizard. For elements created using some method other than the wizard, there may be some differences between the original element and the edited element, so users are requested to examine the differences to ensure that nothing was changed unexpectedly.

For specific details on adding or editing MP components, please see the sections below.

Adding and Editing Folders

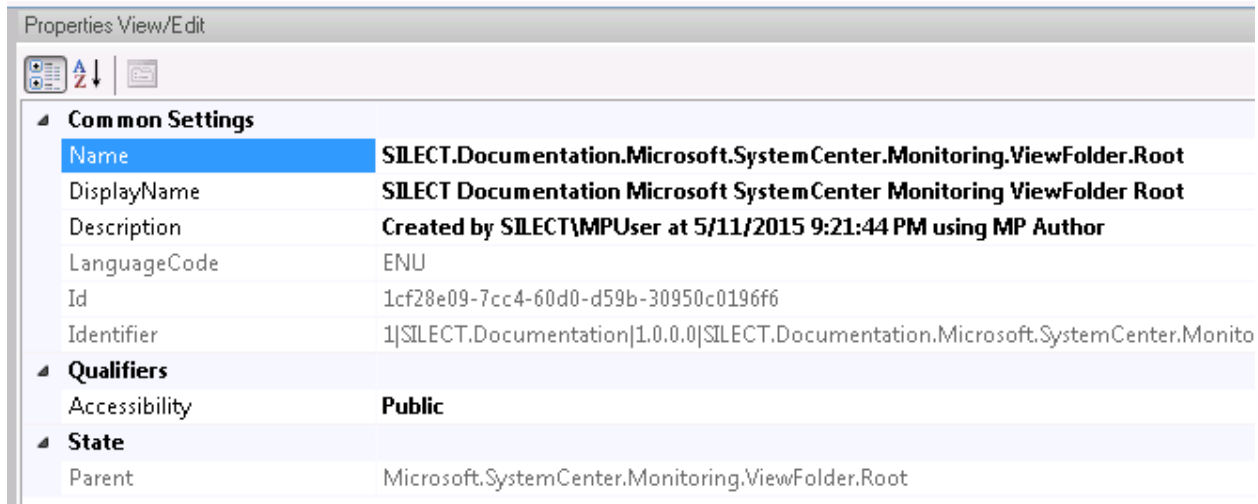
Folders are used by Operations Manager to organize and present information in the Operations Manager Console. To add a new folder to an MP, follow these steps:

- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 When the MP has been opened, right-click the **Folders** container and select **Create New Folder**.
- Step 3 Select the name of the parent folder, or **None**, from the drop-down. You can use the check box to switch between showing display names and formal names. Click **Next**.
- Step 4 Enter the name, display name and description for the new folder. Click **Next**.
- Step 5 Review the settings and click **Finish** to create the new folder.

To save these changes, click the **Save** toolbar button or select **File, Save**.

To edit a folder, follow these steps:

- Step 1 With the MP already opened, click the **Folders** container to view the folders currently defined in the MP.
- Step 2 Right-click one of the folders displayed and select:
 - a. **Edit with Wizard...** to edit with the same wizard used to create the folder,
 - b. **View/Edit XML** to edit the underlying XML,
 - c. **Delete...** to remove the folder,
 - d. **Properties...** to view detailed properties of the folder.
- Step 3 Or click on one of the folders and change the properties which are not greyed out in the **Properties View/Edit** window.



To save changes, click the **Save** toolbar button or select **File, Save**.

Adding and Editing Classes and Discoveries

Discoveries are used by Operations Manager to discover the components of an application or system. Classes are used to describe the attributes of one or more objects (known as instances of the class). A target is an instance of a class. Many class/discovery wizards create both the class and a discovery to create instances of the class. To create a new class (and often the corresponding discovery), follow these steps:

- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 When the MP has been opened, right-click the **Classes** or **Discoveries** container and select one of the available options:
 - Create New Group**
 - Create New Derived Class ***
 - Create New Class *** (under Classes only)
 - Create New Windows Registry Class**
 - Create New Windows Script Class**
 - Create New Windows WMI Class**
 - Create New SNMP Class ***

* Feature available only in Silect MP Studio or MP Author Pro.

Refer to the appropriate section below for details on creating each type of class/discovery.

Creating a New Group

This wizard is used to create a group that can be used to contain multiple items. To create a new group:

- Step 1 Right-click on the **Classes** or **Discoveries** node, and in the popup menu that appears, select the **Create New Group** menu item.
- Step 2 Enter a name, display name and description for the new group. Click **Next**.
- Step 3 (Optional) Specify the membership rules which determine what is included in this new group. You can specify rules which determine that instances of a specified class are included in your group, as well as optional expressions which include only a subset of those class instances. Click **Next**.

Step 4 Review the settings and click **Finish** to complete the creation of the new group.

Creating a New Class

(MP Studio & MP Author Pro only)

This wizard is used to create a new class, without a discovery. This can be used, along with relationships, to create a complex health model. To create a new class:

- Step 1 Right-click on the **Classes** node, and in the popup menu that appears, select the **Create New Class** menu item.
- Step 2 Specify any properties you wish to define for your new class. Click **Next**.
- Step 3 Specify the Base Class for your new class, and specify whether it is to be Abstract, Hosted and/or a Singleton class (these selections determine whether you must later specify a Hosting Class or a Relationship Class). Click **Next**.
- Step 4 If a Hosting Class must be specified, you will be asked to specify it. Click **Next**.
- Step 5 Enter the name, display name and description for the new class. Click **Next**.
- Step 6 If a relationship must be specified for your new class, you will be asked to specify the name, display name and description for the new relationship. Click **Next**.
- Step 7 Review the settings and click **Finish** to complete the creation of the new Class.

Creating a New Derived Class

(MP Studio & MP Author Pro only)

This wizard is used to create a new class, without a discovery, from an existing class. To create a new derived class:

- Step 1 Right-click on the **Classes** or **Discoveries** node, and in the popup menu that appears, select the **Create New Derived Class** menu item.
- Step 2 Specify the target class. You can use a group filter here to create class instances for each member of a group. Click **Next**.
- Step 3 Enter the name, display name and description for the new class. Click **Next**.
- Step 4 Specify the Base Class for your new class. Click **Next**.
- Step 5 Enter a name, display name and description for the new derived discovery. Choose to enable (default) or disable this new derived discovery. Click **Next**.
- Step 6 Specify an optional expression. If specified, then only target class instances where the expression is true will be used to create new derived class instances. Click **Next**.
- Step 7 Enter the discovery schedule. By default, it will run every 4 hours (since it has a very low impact). Click **Next**.
- Step 8 Review the settings and click **Finish** to complete the creation of the new Derived Class.

Creating a New Windows Registry Target

A registry target is used to identify the existence of an application or system using a registry entry or value. To create a new registry target:

- Step 1 Right-click on the **Classes** or **Discoveries** node, and in the popup menu that appears, select the **Create New Windows Registry Class** menu item.
- Step 2 Next you will be prompted to enter the name of a machine to browse the registry to help define the new registry target. Enter the name of a machine or click the **Browse** button to browse the network. Click **Next**.

Tip Browsing a computer where the application or system you want to monitor is installed will help create the objects in the new MP. Credentials used for browsing remote systems are

managed using the **Tools, Manager Server Credentials** dialog.

- Step 3 Click the **Browse** button to open the registry on the machine specified earlier. Navigate thru the registry settings to find the setting or settings you want to include for targets. Add additional mandatory and optional properties as required. Click **Next**.
- Step 4 If optional properties were specified, you may wish to adjust the information about the class properties which will be created. Click **Next**.
- Step 4 Enter a name, display name and description for the new registry class. Click **Next**.
- Step 5 Choose the base class to use for this new class. Click **Next**.
- Step 6 Enter a name, display name and description for the new registry discovery. Choose to enable (default) or disable this new registry discovery. Click **Next**.
- Step 7 Choose the target class to run this discovery against. A group can be specified to filter the target class instances. Click **Next**.
- Step 8 If you have multiple settings, enter the expression you want to use for the target. You can use a logical expression or simply check the existence of a registry key. Click **Next**.
- Step 9 Enter the discovery schedule. By default, it will run once per day. Click **Next**.
- Step 10 Review the settings and click **Finish** to complete the creation of the new registry target.

Creating a New Windows Script Target

A script target is used to identify the existence of an application or system using the results of running a script. To create a new script target:

- Step 1 Right-click on the **Classes** or **Discoveries** node, and in the popup menu that appears, select the **Create New Windows Script Class** menu item.
- Step 2 Enter the name of the script or use the **Browse** button to find the script that will be used to discover the target. The class name and any properties will be parsed from the script.
- Step 3 You can optionally add parameters to the script. Click the **Add** button to add a parameter to the script. Specify the name of the parameter, the initial value (these can be overridden) and the data type. Repeat to include as many parameters as required. For PowerShell scripts, the list of parameters will be already filled in, but you will need to provide default values. Double click on the parameter and enter the parameter value as needed. Click **Next**.
- Step 4 The script will have been parsed to determine the class properties. You can add optional properties for the new class. Click the **Add** button to add a property. Click **Next**.
- Step 5 The script will have been parsed to determine the class name, which cannot be changed. Enter a display name and description for the new script class. Click **Next**.
- Step 6 Choose the base class to use for this new class. Click **Next**.
- Step 7 Enter a name, display name and description for the new script discovery. Choose to enable (default) or disable this new script discovery. Click **Next**.
- Step 8 Choose the target class to run this discovery against. A group can be specified to filter the target class instances. Click **Next**.
- Step 9 Enter the discovery schedule. By default, it will run once per day. Click **Next**.
- Step 10 Review the settings and click **Finish** to complete the creation of the new script target.

Creating a New Windows WMI Target

A WMI target is used to identify the existence of an application or system using information from the WMI database. To create a new WMI target:

- Step 1 Right-click on the **Classes** or **Discoveries** node, and in the popup menu that appears, select the **Create New Windows WMI Class** menu item.

- Step 2 Next you will be prompted to enter the name of a machine to browse the WMI database to help define the new WMI target. Enter the name of a machine or click the **Browse** button to browse the network. Click **Next**.

Tip Browsing a computer where the application or system you want to monitor is installed will help create the objects in the new MP. Credentials used for browsing remote systems are managed using the **Tools, Manager Server Credentials** dialog.

- Step 3 Click the **Browse** button to open the WMI database on the machine specified earlier. Navigate thru the WMI database to find the class you want to use for the WMI target setting.

Note These next steps will help create a WQL statement used for the WMI target. This can be entered manually as well.

- Step 4 Select an appropriate namespace and class in the **Step 1 - Select namespace/class** window. Click the **Retrieve** button to retrieve instances of the selected object.
- Step 5 Select an appropriate record in the **Step 2 - Select a record** window. This will display all property values for the record.
- Step 6 In the **Step 3 - Create query definition** window, check the properties on the left to include them in the **SELECT** clause (to retrieve those values).
- Step 7 Check the properties on the right to include them in the **WHERE** clause, which must uniquely identify a record. Selecting the key properties (🔑) can be very helpful here.
- Step 8 Click the **OK** button.
- Step 9 Click **Next**.
- Step 10 Add optional properties for the new class by clicking **Add**. Click **Next** when you have completed adding optional properties.
- Step 11 Enter a name, display name and description for the new WMI class. Click **Next**.
- Step 12 Choose the base class to use for this new role. Click **Next**.
- Step 13 Enter a name, display name and description for the new WMI discovery. Choose to enable (default) or disable this new WMI discovery. Click **Next**.
- Step 14 Choose the target class to run this discovery against. A group can be specified to filter the target class instances. Click **Next**.
- Step 15 Enter the discovery schedule. By default, it will run once per day. Click **Next**.
- Step 16 Review the settings and click **Finish** to complete the creation of the new WMI target.

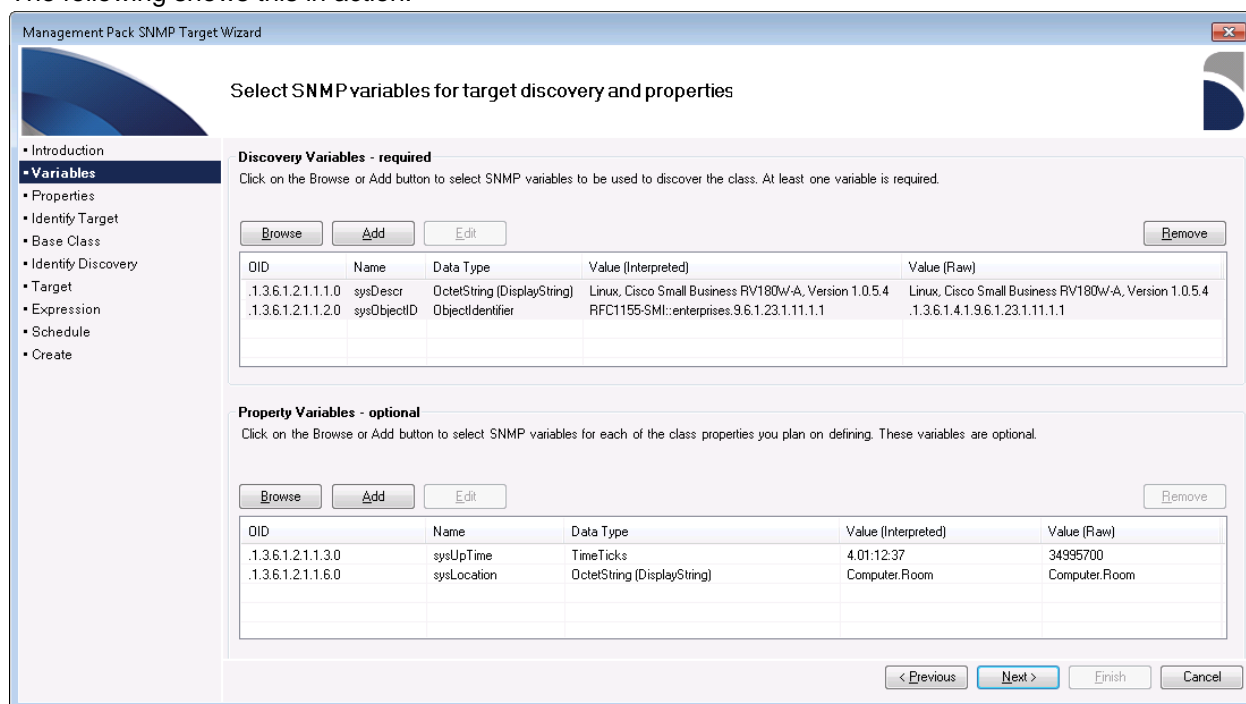
Creating a New SNMP Target

(MP Studio & MP Author Pro only)

An SNMP target is used to identify the existence of a device or computer that communicates using the SNMP protocol. To create a new SNMP target:

- Step 1 Right-click on the **Classes** or **Discoveries** node, and in the popup menu that appears, select the **Create New SNMP Class** menu item.
- Step 2 In the **Introduction** step for the wizard, if you are working on with a new SNMP device, click on the **Browse** button to specify the SNMP device you wish to connect to.
- Step 3 In the **Select the network device to read data from** dialog, specify the IP address of the network device, and the SNMP protocol version and credentials which you wish to use to connect to your device.

- Step 4 Once you have input the required information, click on the **Validate** button to ensure that you have specified the network device information correctly. If you have, the name of the device you connected to should be displayed in the **Name** field (this can be changed, if desired).
- Step 5 Click on the **Save** button to save the connection settings for your SNMP device with the specified name.
- Step 6 Then click on the **OK** button to return to the Introduction step for your wizard. The Introduction step should now have filled in the name of the new device that you just connected to.
- Step 7 Click **Next** to proceed to the **Variables** step.
- Step 8 In the mandatory **Discovery Variables** section click **Browse** to view the SNMP variables returned from the device that can be used for discovery. This will result in the **Add Network Device Variables** dialog appearing.
- Step 9 On the **Add Network Device Variables** dialog, if the MIB files used to decode the SNMP variables are not specific enough to have interpreted all the SNMP variables, a warning will be displayed and the **MIBs** button in the dialog will be displayed in red. Click on it, which will pop up the **Manage MIBs** dialog which can be used to locate and load the correct MIBs for your device (best place to get this will always be from your device manufacturer). Once you have located and installed the correct MIB files, you can click on **OK** to return to the **Add Network Device Variables** dialog.
- Step 10 At this point, on the **Variables** step of the wizard, select SNMP variables from your device to be used to identify the SNMP device you wish to discover (typical variables would be the **sysName** or **sysObjectID** variables). You can also define optional SNMP variables which will be used to set properties for the class created for your SNMP device by the discovery. The following shows this in action:



- Step 11 Click **Next** after you have completed defining your SNMP variables.
- Step 12 On the following **Properties** screen of the wizard, you will be asked to optionally configure any properties for your SNMP discovery. Click on **Next** when you are finished.

- Step 13 On the next Identify Target screen, you will be asked to create a name, display name and description for the class which you will discover. You would typically do this by changing the text in the Name field from «**Name**» to the name of your SNMP device. Once you are finished, you can click on **Next**.
- Step 14 On the next Base class screen, you will be asked to identify the base class of your target class. Click **Next** when you have selected your base class.
- Step 15 On the next **Identify Discovery** screen, you will be given the opportunity to edit the name, display name and description of your discovery. For most users, the default value should be okay. Click **Next** when you are finished.
- Step 16 On the next **Target** screen, you will be given the opportunity to change the target class for your discovery. For most users, the default value of **System.NetworkManagement.Node** should be okay. A group can be specified to filter the target class instances. Click **Next** when you are finished.
- Step 17 On the next **Expression** screen, you are given the opportunity to fine tune the expression used to discover the instances of your SNMP device. When you are happy with the expression, click **Next** to proceed.
- Step 18 On the next **Schedule** screen, you can configure the discovery schedule. By default, it will run once per day. Click **Next**.
- Step 19 Review the settings and click **Finish** to complete the creation of the new SNMP target.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Editing a Class

To edit a class, follow these steps:

- Step 1 With the MP already opened, click the **Classes** container to view the classes currently defined in the MP.
- Step 2 Right-click one of the classes displayed and select:
- Edit with Wizard...** to edit with the same wizard used to create the class (and usually the discovery),
 - View/Edit XML** to edit the underlying XML,
 - Delete...** to remove the class,
 - Properties...** to view detailed properties of the class.
- Step 3 Or click on one of the classes and change the properties which are not greyed out in the **Properties View/Edit** window.

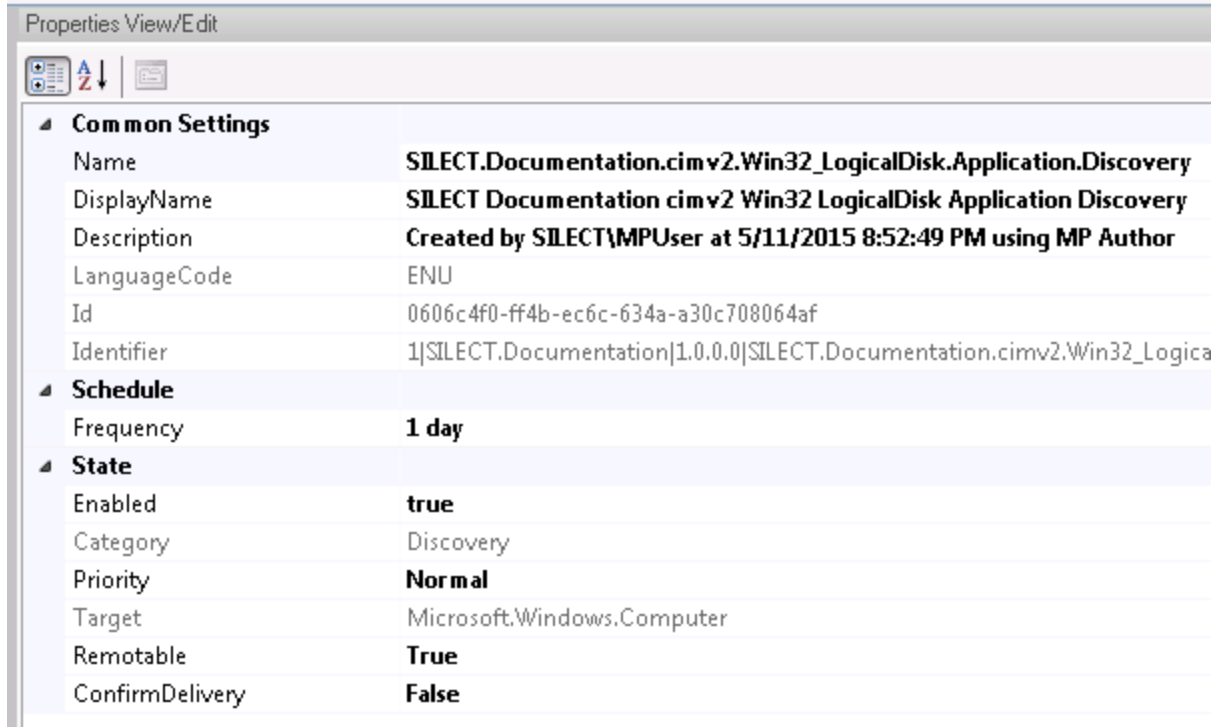
Properties View/Edit	
Common Settings	
Name	SILECT.Documentation.Application.Rollup
DisplayName	SILECT Documentation Application Rollup
Description	Created by SILECT\MPUser at 5/11/2015 8:26:27 PM using MP Author
LanguageCode	ENU
Id	957a5974-60d4-e849-cb0d-eda40fecb626
Identifier	1 SILECT.Documentation 1.0.0.0 SILECT.Documentation.Application.Rollup
Qualifiers	
Abstract	False
Hosted	False
Singleton	False
Extension	False
Base	System.Service
Accessibility	Public

To save these changes, click the **Save** toolbar button or select **File, Save**.

Editing Discoveries

Discoveries are used by Operations Manager to discover things to manage. To edit Discoveries in an MP, follow these steps:

- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 When the MP has been opened, click the **Discoveries** container to view the discoveries currently defined within the MP.
- Step 3 Right-click one of the discoveries displayed and select:
 - a. **Edit with Wizard...** to edit with the same wizard used to create the class and discovery,
 - b. **View/Edit XML** to edit the underlying XML,
 - c. **Create/Edit Knowledge** to edit the knowledge associated with the discovery,
 - d. **Delete...** to remove the discovery,
 - e. **Properties...** to view detailed properties of the discovery.
- Step 4 Or click on one of the discoveries and change the properties which are not greyed out in the **Properties View/Edit** window.



To save these changes, click the **Save** toolbar button or select **File, Save**.

Adding and Editing Rules

Rules are used by Operations Manager to collect information about components of an application or system. To add a new rule to an existing MP, follow these steps:

- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 When the MP has been opened, right-click the **Rules** container and select one of the available options:
 - Create New Windows Event/Alert Rule**
 - Create New Windows Performance Rule**
 - Create New Windows Script Performance Rule**
 - Create New SNMP Probe Event Rule ***
 - Create New SNMP Probe Performance Rule ***
 - Create New SNMP Trap Event/Alert Rule ***

* Feature available only in Silect MP Studio and MP Author Pro.

Refer to the appropriate section below for details on creating each type of rule.

Creating an Event/Alert Rule

An event rule is used to collect information from the Windows Event Log. To create a new event rule with an optional alert, follow these steps:

- Step 1 Right-click on the **Rules** node, and in the popup menu that appears, select the **Create New Windows Event/Alert Rule** menu item.

- Step 2 Enter the name of a machine to browse the Event Log to help define the new event rule. Or click the **Browse** button to browse the network. Click **Next**.

Tip Browsing a computer where the application or system you want to monitor is installed will help create the objects in the new MP. Credentials used for browsing remote systems are managed using the **Tools, Manager Server Credentials** dialog.

- Step 3 Select the **Event Log** for this rule from the drop-down list of event logs. Next enter the expression to evaluate event log entries. By default, you will be prompted for an Event ID number and optionally an Event Source, but you can select any other properties of the event by clicking on the drop down. Specify if the rule is to collect data (otherwise you will be required to generate an alert). Click **Next**.
- Step 4 Select the **Target** for this rule from the drop-down list of targets. Optionally, filter the target by a group (suitable overrides will be created). Click **Next**.
- Step 5 Enter a name, display name and description for the new event rule. Click **Next**.
- Step 6 Optionally choose to create an Alert for this monitor including Priority, Severity and Message text. In Step 2, if you chose to not collect data, then this is required. Click **Next**.
- Step 7 Specify the schedule to run event rule. Click **Next**.
- Step 8 Review the settings and click **Finish** to complete the creation of the new event rule.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New Windows Performance Rule

A performance rule is used to collect Windows Performance information. To create a new performance rule:

- Step 1 Right-click on the **Rules** node, and in the popup menu that appears, select the **Create New Windows Performance Rule** menu item.
- Step 2 Enter the name of a machine to browse the Windows Performance Monitor to help define the new performance rule. Or click the **Browse** button to browse the network. Click **Next**.

Tip Browsing a computer where the application or system you want to monitor is installed will help create the objects in the new MP. Credentials used for browsing remote systems are managed using the **Tools, Manager Server Credentials** dialog.

- Step 3 Select the Windows Performance Monitor **Object, Counter, Instance** and **Sampling Rate** to define this rule. Click **Next**.
- Step 4 Select the **Target** for this rule from the drop-down list of targets. Optionally, filter the target by a group (suitable overrides will be created). Click **Next**.
- Step 5 Enter a name, display name and description for the new performance rule. Click **Next**.
- Step 6 Optionally create a view for this rule, and/or save to the data warehouse. Click **Next**.
- Step 7 Specify the schedule to run performance rule. Click **Next**.
- Step 8 Review the settings and click **Finish** to complete the creation of the new performance rule.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New Windows Script Performance Rule

A script performance rule is used to capture performance information using a PowerShell script file. To create a new script performance rule:

- Step 1 Right-click on the **Rules** node, and in the popup menu that appears, select the **Create New Windows Script Performance Rule** menu item.
- Step 2 Select the **Target** for this rule from the drop-down list of targets. Optionally, filter the target by a group (suitable overrides will be created). Click **Next**.
- Step 3 Enter the name of the script or use the **Browse** button to find the script that will be used to collect the performance data. You can optionally add parameters to the script. Click the **Add** button to add a parameter to the script. Specify the name of the parameter, the initial value (these can be overridden) and the data type. Repeat to include as many parameters as required. For PowerShell scripts, the list of parameters will be already filled in, but you will need to provide default values. Double click on the parameter and enter the parameter value as needed. Click **Next**.
- Step 4 Enter the **Object, Counter, Instance** and **Value** mappings. A list of properties returned in the property bag is displayed to make it easier to specify property names. Click **Next**.
- Step 5 Enter a name, display name and description for the new script rule. Click **Next**.
- Step 6 Optionally create a view for this rule, and/or save to the data warehouse. Click **Next**.
- Step 7 Specify the schedule to run the script rule. Click **Next**.
- Step 8 Review the settings and click **Finish** to complete the creation of the new script performance rule.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New SNMP Probe Event Rule

(MP Studio & MP Author Pro only)

An SNMP probe event rule is used to capture event type information from an SNMP device using an SNMP probe. To create a new SNMP probe event rule:

- Step 1 Right-click on the **Rules** node, and in the popup menu that appears, select the **Create New SNMP Probe Event Rule** menu item.
- Step 2 Enter the name of the previously defined SNMP device to browse its SNMP variables to help define the new event rule. Or click the **Browse** button to browse the network and define a new SNMP device to collect variables from. Click **Next**.
- Step 3 Use the **Browse** or **Add** buttons on this dialog to add the SNMP variables to choose to identify events. Click **Next**.
- Step 4 Specify the configuration for the event information that you will be collecting from the SNMP device. Click **Next**.
- Step 5 Select the target class for the rule. Optionally, filter the target by a group (suitable overrides will be created). Click **Next**.
- Step 6 Enter a name, display name and description for the new SNMP event rule. Click **Next**.
- Step 7 Specify the schedule to run the event rule. Click **Next**.
- Step 8 Review the settings and click **Finish** to complete the creation of the new event rule.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New SNMP Probe Performance Rule

(MP Studio & MP Author Pro only)

An SNMP probe performance rule is used to capture performance information from an SNMP device using an SNMP probe. To create a new SNMP probe performance rule:

- Step 1 Right-click on the **Rules** node, and in the popup menu that appears, select the **Create New SNMP Probe Performance Rule** menu item.
- Step 2 Enter the name of the previously defined SNMP device to browse its SNMP variables to help define the new performance rule. Or click the **Browse** button to browse the network and define a new SNMP device to collect variables from. Click **Next**.
- Step 3 Use the **Browse** or **Add** buttons on this dialog to add SNMP variables to probe for performance information. Multiple variables can be collected and used in an expression in Step 8 below. Click **Next**.
- Step 4 Enter the configuration information for the rule. Click **Next**.
- Step 5 Select the target class for the rule. Optionally, filter the target by a group (suitable overrides will be created). Click **Next**.
- Step 6 Enter a name, display name and description for the new SNMP probe performance rule. Click **Next**.
- Step 7 Specify the schedule to run the event rule. Click **Next**.
- Step 8 Optionally add an expression performing numeric calculations on the variables specified in Step 2. Click **Next**.
- Step 9 Review the settings and click **Finish** to complete the creation of the new SNMP probe performance rule.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New SNMP Trap Event/Alert Rule

(MP Studio & MP Author Pro only)

An SNMP trap event/alert rule is used to create event type information from an SNMP device using a trap. An optional alert can also be created based on the occurrence of the trap. To create a new SNMP trap event/alert rule:

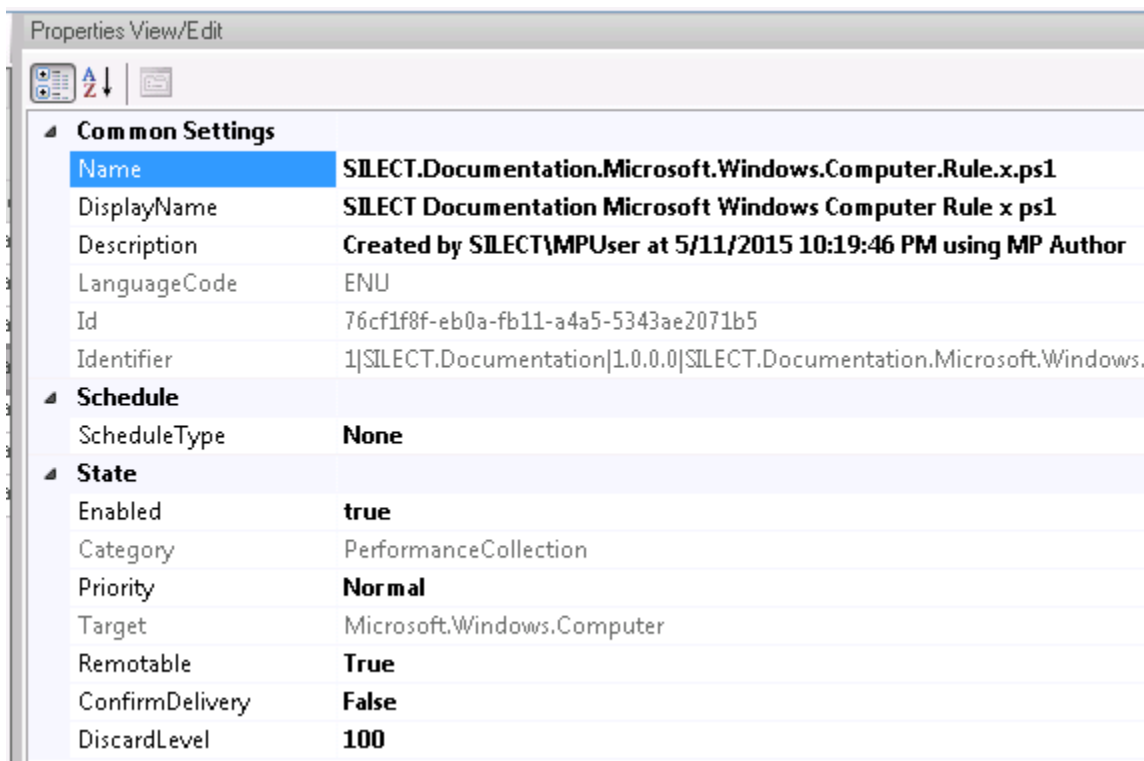
- Step 1 Right-click on the **Rules** node, and in the popup menu that appears, select the **Create New SNMP Trap Event/Alert Rule** menu item.
- Step 2 Enter the name of the previously defined SNMP device to browse its SNMP traps to help define the new SNMP trap event/alert rule. Or click the **Browse** button to browse the network and define a new SNMP device to collect traps from. Click **Next**.
- Step 3 Use the **Browse** or **Add** buttons on this dialog to add the SNMP traps to associate with your event. When you are finished adding traps to associate with your event, click **Next**.
- Step 4 Specify the configuration for the event information that you will be generating for the SNMP device when the trap occurs. Click **Next**.
- Step 5 Select the target class for the rule. Optionally, filter the target by a group (suitable overrides will be created). Click **Next**.
- Step 6 You can optionally define an expression that would be used to filter the traps. Click **Next**.
- Step 7 Enter a name, display name and description for the new SNMP trap event/alert rule. Click **Next**.
- Step 8 Optionally specify the alert to be raised when the trap event occurs. Click **Next**.
- Step 9 Review the settings and click **Finish** to complete the creation of the new SNMP trap event/alert rule.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Editing Rules

To edit a Rule, follow these steps:

- Step 1 With the MP already opened, click the **Rules** container to view the rules currently defined in the MP.
- Step 2 Right-click one of the rules displayed and select:
 - a. **Edit with Wizard...** to edit with the same wizard used to create the rule,
 - b. **View/Edit XML** to edit the underlying XML,
 - c. **Create/Edit Knowledge** to edit the knowledge associated with the rule,
 - d. **Delete...** to remove the rule,
 - e. **Properties...** to view detailed properties of the rule.
- Step 3 Or click on one of the rules and change the properties which are not greyed out in the **Properties View/Edit** window.



To save these changes, click the **Save** toolbar button or select **File, Save**.

Adding and Editing Monitors

Monitors are used by Operations Manager to monitor the state or conditions of an application or system.

To add a new monitor to an existing MP, follow these steps:

- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 When the MP has been opened, right-click the **Monitors** container and select one of the available options:
 - Create New Windows Database Monitor**
 - Create New Windows Event Monitor**
 - Create New Windows Performance Monitor**

- Create New Windows Process Monitor ***
- Create New Windows Script Monitor**
- Create New Windows Service Monitor**
- Create New Windows Web Site Monitor**
- Create New SNMP 2-State 2-Probe Monitor ***
- Create New SNMP 2-State 2-Trap Monitor ***
- Create New Dependency Rollup Monitor ***
- Create New Aggregate Rollup Monitor ***

* Feature available only in Silect MP Studio and MP Author Pro.

Refer to the appropriate section below for details on creating each type of monitor.

Creating a New Windows Database Monitor

A database monitor is used to determine the health state of a database component either on its own or part of a larger application or system. To create a new database monitor:

- Step 1 Right-click on **Monitors** and select **Create New Windows Database Monitor**.
- Step 2 Next you will be prompted to enter a database connection string. Click **Edit** to browse and edit SQL connection information. If you want to run a SQL query as part of this monitor, click the **Query String to Execute (Optional)** check box and enter an appropriate SQL statement. Next specify the SQL timeout value. Click **Next**.

Tip Browsing a computer where the application or system you want to monitor is installed will help create the objects in the new MP. Credentials used for browsing remote systems are managed using the **Tools, Manager Server Credentials** dialog.

- Step 3 Select the **Polling Interval** to determine the frequency of polling the monitor. Select **Connection Time** warning and error thresholds. If you have specified a SQL query as part of this monitor, specify the **Query Time** and **Fetch Time** thresholds. Click **Next**.
- Step 4 Enter a display name for the new database monitor (the template does not allow a name to be specified). Click **Next**.
- Step 5 Specify the **Run As** credentials to be used by this monitor or leave **(none / default)** specified to use the default credentials. Click **Next**.
- Step 6 Specify the **Watcher Nodes** for this monitor. You can use the default machine name or browse for other watcher nodes. Click **Next**.
- Step 7 Review the settings and click **Finish** to complete the creation of the new database monitor.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New Windows Event Monitor

An event monitor is used to determine the health state of an application or system based on Event Log entries. To create a new event monitor:

- Step 1 Right-click on **Monitors** and select **Create New Windows Event Monitor**.
- Step 2 Next you will be prompted to enter the name of a machine to browse the Event Log to help define the new event monitor. Enter the name of a machine or click the **Browse** button to browse the network. Click **Next**.

Tip Browsing a computer where the application or system you want to monitor is installed will

help create the objects in the new MP. Credentials used for browsing remote systems are managed using the **Tools, Manager Server Credentials** dialog.

- Step 3 Select the **Event Log** for this Monitor from the drop-down list of Event Logs. Next enter the expression to evaluate event log entries. By default, you will be prompted for an Event ID number and optionally an Event Source, but you can choose any property of the event by clicking the drop down. Click **Next**.
- Step 4 Select the **Target** for this monitor from the drop-down list of targets. Optionally, filter the target by a group (suitable overrides will be created). Select the **Parent Monitor** from the list of parent monitors. Click **Next**.
- Step 5 Specify either a wait time before resetting the health state of this monitor or specify an event used to reset the health state. Click **Next**.
- Step 6 Enter a name, display name and description for the new event monitor. Click **Next**.
- Step 7 Specify the **Run As** credentials to be used by this monitor or leave (**none / default**) specified to use the default credentials. Click **Next**.
- Step 8 Specify the health state for each of the detected conditions. Click **Next**.
- Step 9 Optionally choose to create an Alert for this monitor including Priority, Severity and Message text. Click **Next**.
- Step 10 Review the settings and click **Finish** to complete the creation of the new event monitor.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New Windows Performance Monitor

A performance monitor is used to determine the health state of an application or system based on Windows performance information. To create a new performance monitor:

- Step 1 Right-click on **Monitors** and select **Create New Windows Performance Monitor**.
- Step 2 Next you will be prompted to enter the name of a machine to browse the Windows Performance Monitor to help define the new performance monitor. Enter the name of a machine or click the **Browse** button to browse the network. Click **Next**.

Tip Browsing a computer where the application or system you want to monitor is installed will help create the objects in the new MP. Credentials used for browsing remote systems are managed using the **Tools, Manager Server Credentials** dialog.

- Step 3 Select the Windows Performance Monitor **Object, Counter, Instance, Number of Samples** and **Sampling Rate** to define this monitor. Optionally create a rule to collect the performance data. Click **Next**.
- Step 4 Select the **Target** for this monitor from the drop-down list of targets. Optionally, filter the target by a group (suitable overrides will be created). Select the **Parent Monitor** from the list of parent monitors. Click **Next**.
- Step 5 Enter a name, display name and description for the new performance monitor. Click **Next**.
- Step 6 Specify the **Run As** credentials to be used by this monitor or leave (**none / default**) specified to use the default credentials. Click **Next**.
- Step 7 Specify the type of monitor, condition to test, number of samples and thresholds for the monitor. Click **Next**.

- Step 8 Optionally choose to create an Alert for this monitor including Priority, Severity and Message text. Click **Next**.
- Step 9 Review the settings and click **Finish** to complete the creation of the new performance monitor.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New Windows Process Monitor

(MP Studio & MP Author Pro only)

A process monitor is used to determine the health state of an application or system based on whether a process is running or not running. To create a new process monitor:

- Step 1 Right-click on Monitors and select **Create New Windows Process Monitor**.
- Step 2 Next you will be prompted to enter the name of a machine to browse the processes to help define the new process monitor. Enter the name of a machine or click the **Browse** button to browse the network. Click **Next**.

Tip Browsing a computer where the application or system you want to monitor is installed will help create the objects in the new MP. Credentials used for browsing remote systems are managed using the **Tools, Manager Server Credentials** dialog.

- Step 3 Select the process to be monitored. Click **Next**.
- Step 4 Select the **Target** for this monitor from the drop-down list of targets. Optionally, filter the target by a group (suitable overrides will be created). Select the **Parent Monitor** from the list of parent monitors. Click **Next**.
- Step 5 Specify whether the process is to be running, and the number of processes, or whether it must not be running. Click **Next**.
- Step 6 Enter a name, display name and description for the new process monitor. Click **Next**.
- Step 7 Specify the **Run As** credentials to be used by this monitor or leave **(none / default)** specified to use the default credentials. Click **Next**.
- Step 8 Optionally choose to create an Alert for this monitor including Priority, Severity and Message text. Click **Next**.
- Step 9 Review the settings and click **Finish** to complete the creation of the new process monitor.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New Windows Script Monitor

A script monitor is used to determine the health state of an application or system based on the results of running a script. To create a new script monitor:

- Step 1 Right-click on **Monitors** and select **Create New Windows Script Monitor**.
- Step 2 Select the **Target** for this monitor from the drop-down list of targets. Optionally, filter the target by a group (suitable overrides will be created). Select the **Parent Monitor** from the list of parent monitors. Click **Next**.
- Step 3 Enter the name of the script or use the **Browse** button to find the script that will be used to determine the health state. You can optionally add parameters for the script. For PowerShell scripts, the list of parameters will be already filled in, but you will need to provide default values. Double click on the parameter and enter the parameter value as needed. Click the **Add** button to add more parameters for the script, if needed. Specify the name of the parameter, the initial value (these can be overridden) and the data type. Repeat to include as many parameters as required. Click **Next**.

- Step 4 Select monitor states and criteria. For each criterion, replace «**Name**» with the name of a value returned in the property bag from the script and set the condition. A list of properties returned in the property bag is displayed to make it easier to specify property names. Click **Next**.
- Step 5 Enter a name, display name and description for the new script monitor. Click **Next**.
- Step 6 Specify the **Run As** credentials to be used by this monitor or leave (**none / default**) specified to use the default credentials. Click **Next**.
- Step 7 Specify the schedule to run the script monitor. Click **Next**.
- Step 8 Optionally choose to create an Alert for this monitor including Priority, Severity and Message text. Click **Next**.
- Step 9 Review the settings and click **Finish** to complete the creation of the new script monitor.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New Windows Service Monitor

A service monitor is used to determine the health state of an application or system based on the state (running or not running) of a Windows service. To create a new service monitor:

- Step 1 Right-click on **Monitors** and select **Create New Windows Service Monitor**.
- Step 2 Next you will be prompted to enter the name of a machine to browse installed services to help define the new service monitor. Enter the name of a machine or click the **Browse** button to browse the network. Click **Next**.

Tip Browsing a computer where the application or system you want to monitor is installed will help create the objects in the new MP. Credentials used for browsing remote systems are managed using the **Tools, Manager Server Credentials** dialog.

- Step 3 Select the Windows Service to monitor. Click **Next**.
- Step 4 Select the **Target** for this monitor from the drop-down list of targets. Optionally, filter the target by a group (suitable overrides will be created). Select the **Parent Monitor** from the list of parent monitors. Click **Next**.
- Step 5 Enter a name, display name and description for the new service monitor. Click **Next**.
- Step 6 Specify the **Run As** credentials to be used by this monitor or leave (**none / default**) specified to use the default credentials. Click **Next**.
- Step 7 Specify the health states for the various service states. Click **Next**.
- Step 8 Optionally choose to create an Alert for this monitor including Priority, Severity and Message text. Click **Next**.
- Step 9 Review the settings and click **Finish** to complete the creation of the new service monitor.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New Windows Web Site Monitor

A web site monitor is used to determine the health state of a web service component either on its own or part of a larger application or system. To create a new web site monitor:

- Step 1 Right-click on **Monitors** and select **Create New Windows Web Site Monitor**.
- Step 2 Next you will be prompted to enter the URL of the web server to monitor and polling interval. Click **Next**.
- Step 3 Enter a display name for the new web site monitor (the template does not allow a name to be specified). Click **Next**.

- Step 4 Specify the **Watcher Nodes** for this monitor. You can use the default machine name or browse for other watcher nodes. Click **Next**.
- Step 5 Review the settings and click **Finish** to complete the creation of the new web site monitor.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New Dependency Rollup Monitor

(MP Studio & MP Author Pro only)

A dependency rollup monitor is used to roll up the health state of an application or system from low level components to a higher-level component. To create a new dependency rollup monitor:

- Step 1 Right-click on **Monitors** and select **Create New Dependency Rollup Monitor**.
- Step 2 Select which health categories you wish to create dependency rollup monitors for. More than one can be selected. Select the rollup policy (**Worst Of** is most common). Click **Next**.
- Step 3 Select the **Relationship** from the drop-down list of relationships. The relationship determines what rolls up to what. Click **Next**.
- Step 4 Enter a name, display name and description for the new dependency rollup monitor. Names will have the category added automatically (if not present). Click **Next**.
- Step 5 Optionally choose to create an Alert for this monitor including Priority, Severity and Message text. Click **Next**.
- Step 6 Review the settings and click **Finish** to complete the creation of the new dependency rollup monitor. If more than one category was selected, a dependency rollup monitor will be created for each category.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New Aggregate Rollup Monitor

(MP Studio & MP Author Pro only)

An aggregate rollup monitor is used to roll up the health state of several unit monitors into a single parent monitor. To create a new aggregate rollup monitor:

- Step 1 Right-click on **Monitors** and select **Create New Aggregate Rollup Monitor**.
- Step 2 Select which health categories you wish to create aggregate rollup monitors for. More than one can be selected. The categories are also used to select which unit monitors can be rolled up by the wizard. Select the rollup policy (**Worst Of** is most common). Click **Next**.
- Step 3 Select the **Target** from the drop-down list of targets. The target is also used to select which unit monitors can be rolled up by the wizard. Click **Next**.
- Step 4 Enter a name, display name and description for the new aggregate rollup monitor. Names will have the category added automatically (if not present). Click **Next**.
- Step 5 Optionally choose to create an Alert for this monitor including Priority, Severity and Message text. Click **Next**.
- Step 6 If the current MP has unit monitors with the same target, and of any of the selected categories, the wizard will present a list of eligible unit monitors. You may select (check) those that you want to be re-targeted to the new aggregate monitor. You may also choose to have the alerts from the unit monitors disabled so the only alerts are raised by the aggregate monitor.
- Step 7 Review the settings and click **Finish** to complete the creation of the new aggregate rollup monitor. If more than one category was selected, an aggregate rollup monitor will be created for each category. Any selected unit monitors will also be re-targeted to the new aggregate rollup monitor and alerts will be disabled (if requested).

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New SNMP 2-State 2-Probe Monitor

(MP Studio & MP Author Pro only)

An SNMP 2-state 2-probe monitor is used to determine the health state of a device based on probing monitor variables. To create a new SNMP 2-state 2-probe monitor:

- Step 1 Right-click on **Monitors** and select **Create New SNMP 2-State 2-Probe Monitor**.
- Step 2 Enter the name of the previously defined SNMP device to browse its SNMP variables to help define the new probe monitor. Or click the **Browse** button to browse the network and define a new SNMP device to collect variables from. Click **Next**.
- Step 3 Use the **Browse** or **Add** buttons on this page to add the SNMP variables to identify the first and second (optional) probe monitor events. Click **Next**.
- Step 4 Select the **Target** for this monitor from the drop-down list of targets monitor (the default is typically okay). Optionally, filter the target by a group (suitable overrides will be created). Select the **Parent Monitor** from the list of parent monitors. Click **Next**.
- Step 5 Enter a name, display name and description for the new SNMP probe monitor. Click **Next**.
- Step 6 Specify the **Run As** credentials to be used by this monitor or leave (**none / default**) specified to use the default credentials. Click **Next**.
- Step 7 Specify the schedule to run the SNMP probe monitor. Click **Next**.
- Step 8 Specify the expression used to define the first SNMP probe to trigger on. Click **Next**.
- Step 9 Specify the expression used to define the second SNMP probe to trigger on. Click **Next**.
- Step 10 Specify the health state for both generated conditions for the SNMP probes. Click **Next**.
- Step 11 Optionally choose to create an Alert for this monitor including Priority, Severity and Message text. Click **Next**.
- Step 12 Review the settings and click **Finish** to complete the creation of the new SNMP probe monitor.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New SNMP 2-State 2-Trap Monitor

(MP Studio & MP Author Pro only)

An SNMP 2-state 2-trap monitor is used to determine the health state of a device based on trap events that occur. To create a new SNMP 2-state 2-trap monitor:

- Step 1 Right-click on **Monitors** and select **Create New SNMP 2-State 2-Trap Monitor**.
- Step 2 Enter the name of the previously defined SNMP device to browse its SNMP traps to help define the new trap monitor. Or click the **Browse** button to browse the network and define a new SNMP device to collect variables from. Click **Next**.
- Step 3 Use the **Browse** or **Add** buttons on this dialog to add the SNMP traps to monitor. Click **Next**.
- Step 4 Select the **Target** for this monitor from the drop-down list of target classes. Optionally, filter the target by a group (suitable overrides will be created). Select the **Parent Monitor** from the list of parent monitors. Click **Next**.
- Step 5 Enter a name, display name and description for the new SNMP trap monitor. Click **Next**.
- Step 6 Specify the **Run As** credentials to be used by this monitor or leave (**none / default**) specified to use the default credentials. Click **Next**.
- Step 7 Specify the expression used to define the first SNMP trap to trigger on. Click **Next**.
- Step 8 Specify the expression used to define the second SNMP trap to trigger on. Click **Next**.
- Step 9 Specify the health state for both generated conditions for the SNMP traps. Click **Next**.

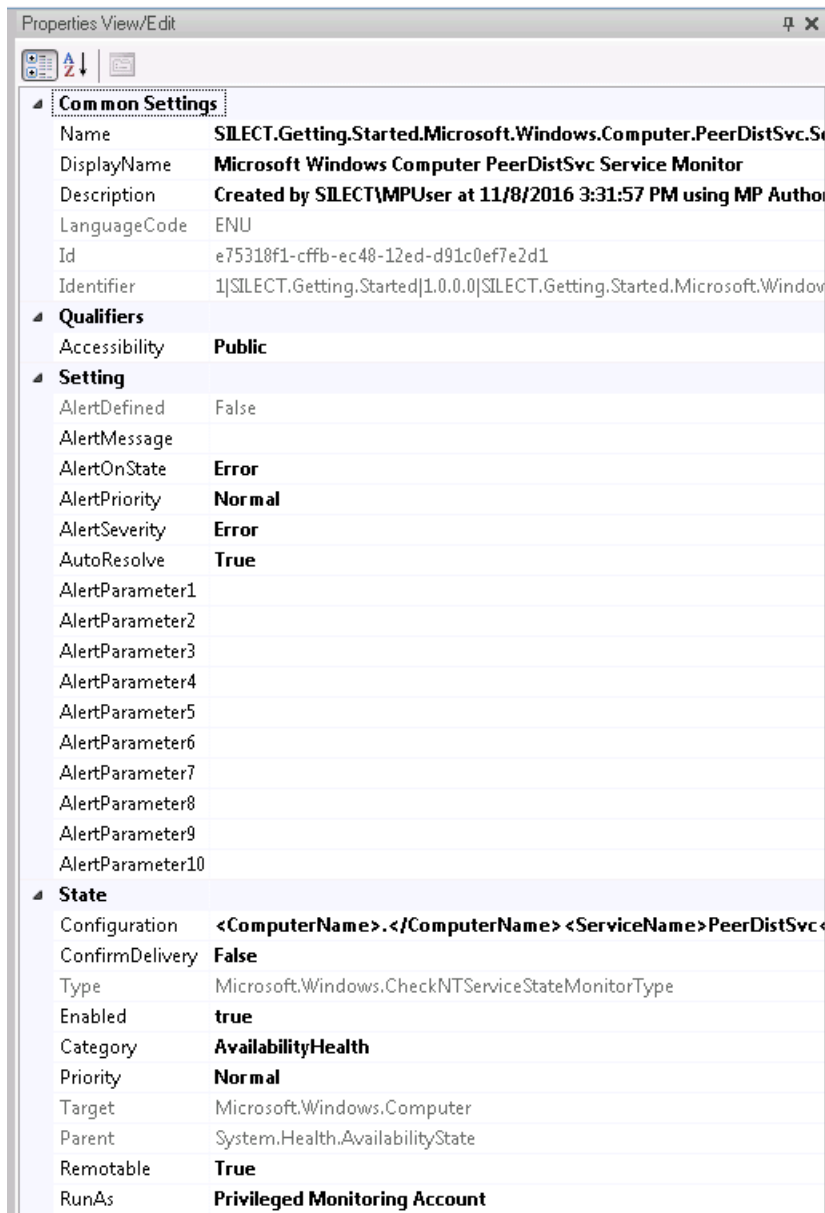
- Step 10 Optionally choose to create an Alert for this monitor including Priority, Severity and Message text. Click **Next**.
- Step 11 Review the settings and click **Finish** to complete the creation of the new SNMP trap monitor.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Editing a Monitor

To edit a monitor, follow these steps:

- Step 1 With the MP already opened, click the **Monitors** container to view the monitors currently defined in the MP.
- Step 2 Right-click one of the monitors displayed and select:
 - a. **Edit with Wizard...** to edit with the same wizard used to create the monitor,
 - b. **View/Edit XML** to edit the underlying XML,
 - c. **Create/Edit Knowledge** to edit the knowledge associated with the monitor,
 - d. **Delete...** to remove the monitor,
 - e. **Properties...** to view detailed properties of the monitor.
- Step 3 Or click on one of the monitors and change the properties which are not greyed out in the **Properties View/Edit** window.



To save these changes, click the **Save** toolbar button or select **File, Save**.

Adding and Editing Views

Views are used by Operations Manager to surface information about the health or operation of an application or system. To add a new view to an existing MP, follow these steps:

- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 When the MP has been opened, right-click the **Views** container and select one of the available options:
 - Create New Alert View**
 - Create New Dashboard View**
 - Create New Event View**

Create New Performance View

Create New State View

Refer to the appropriate section below for details on creating each type of View.

Creating a New Alert View

An alert view provides information about alerts within the Operations Manager environment. To create a new alert view:

- Step 1 Right-click on **Views** and select **Create New Alert View**.
- Step 2 Specify the **Folder** name and the **Category** for the new alert view. Click **Next**.
- Step 3 Select the **Severity** from the drop-down list of severities.
Select the **Priority** from the drop-down list of priorities.
Select the **Resolution State** from the drop-down list.
Select the **Alert Source** from the drop-down list of available alert sources.
Click **Next**.
- Step 4 Select the **Target** for this view from the drop-down list of targets. Specifying a **Group Filter** will restrict the view to only show the members of the specified group. Click **Next**.
- Step 5 Enter a name, display name and description for the new alert view. Click **Next**.
- Step 6 Review the settings and click **Finish** to complete the creation of the new alert view.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New Dashboard View

A dashboard view provides a single pane to look at several other views. To create a new dashboard view:

- Step 1 Right-click on **Views** and select **Create New Dashboard View**.
- Step 2 Specify the **Folder** name and the **Category** for the new dashboard view. Click **Next**.
- Step 3 Select the **Dashboard Configuration**. Choices include anything from a 1 x 1 dashboard grid to a 3 x 3 grid.
Then click each empty cell in the grid and select the **View** to be displayed in that cell from the drop-down list of views. If the same view name is specified in two or more adjacent cells (horizontally or vertically), then the view will span those columns or rows.
Click **Next**.
- Step 4 Select the **Target** for this view from the drop-down list of targets. Click **Next**.
- Step 5 Enter a name, display name and description for the new dashboard view. Click **Next**.
- Step 6 Review the settings and click **Finish** to complete the creation of the new dashboard view.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New Event View

An event view provides information about events within the Operations Manager environment. To create a new event view:

- Step 1 Right-click on **Views** and select **Create New Event View**.
- Step 2 Specify the name of the computer to browse for events as well as the **Folder** name and the **Category** for the new event view. Click **Next**.

Tip Browsing a computer where the application or system you want to monitor is installed will help create the objects in the new MP. Credentials used for browsing remote systems are managed using the **Tools, Manager Server Credentials** dialog.

- Step 3 Add the Event IDs to be included in the new event view.
Or:
Select an event rule to be used as the source for the view.
Click **Next**.
- Step 4 Select the **Target** for this view from the drop-down list of targets. Specifying a **Group Filter** will restrict the view to only show the members of the specified group. Click **Next**.
- Step 5 Enter a name, display name and description for the new event view. Click **Next**.
- Step 6 Review the settings and click **Finish** to complete the creation of the new event view.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New Performance View

A performance view surfaces information about performance data from within the Operations Manager environment. To create a new performance view:

- Step 1 Right-click on **Views** and select **Create New Performance View**.
- Step 2 Specify the name of the computer to browse for performance data as well as the **Folder** name and the **Category** for the new event view. Click **Next**.

Tip Browsing a computer where the application or system you want to monitor is installed will help create the objects in the new MP. Credentials used for browsing remote systems are managed using the **Tools, Manager Server Credentials** dialog.

- Step 3 Select one or more of the following: **Rule, Object, Counter** and **Instance**. Click **Next**.
- Step 4 Select the **Target** for this view from the drop-down list of targets. Specifying a **Group Filter** will restrict the view to only show the members of the specified group. Click **Next**.
- Step 5 Enter a name, display name and description for the new performance view. Click **Next**.
- Step 6 Review the settings and click **Finish** to complete the creation of the new performance view.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating a New State View

A state view surfaces information about state data within the Operations Manager environment. To create a new state view:

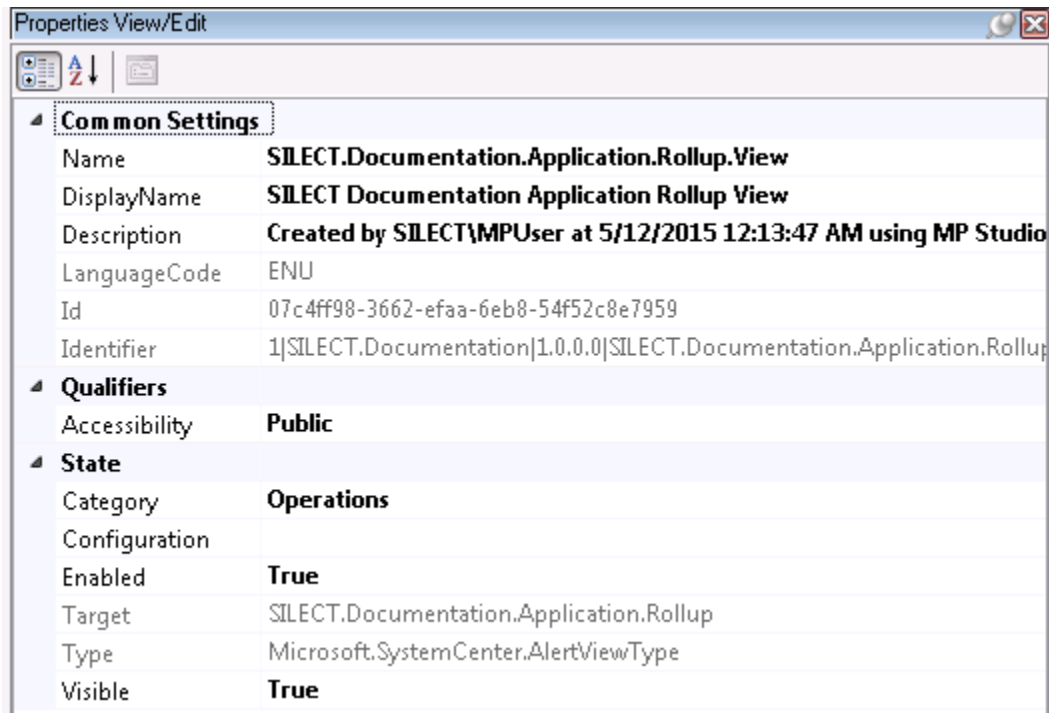
- Step 1 Right-click on **Views** and select **Create New State View**.
- Step 2 Specify the **Folder** name and the **Category** for the new state view. Click **Next**.
- Step 3 Select one or more **Severities** from the list.
Select the **Maintenance Modes** to report.
Optionally, specify a filter with wildcards to only display matching display names.
Click **Next**.
- Step 4 Select the **Target** for this view from the drop-down list of targets. Specifying a **Group Filter** will restrict the view to only show the members of the specified group. Click **Next**.
- Step 5 Enter a name, display name and description for the new state view. Click **Next**.
- Step 6 Review the settings and click **Finish** to complete the creation of the new state view.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Editing Views

To edit a view, follow these steps:

- Step 1 With the MP already opened, click the **View** container to view the views currently defined in the MP.
- Step 2 Right-click one of the views displayed and select:
- Edit with Wizard...** to edit with the same wizard used to create the view,
 - View/Edit XML** to edit the underlying XML,
 - Delete...** to remove the view,
 - Properties...** to view detailed properties of the view.
- Step 3 Or click on one of the views and change the properties which are not greyed out in the **Properties View/Edit** window.



To save these changes, click the **Save** toolbar button or select **File, Save**.

Adding and Editing Relationships

(MP Studio & MP Author Pro only)

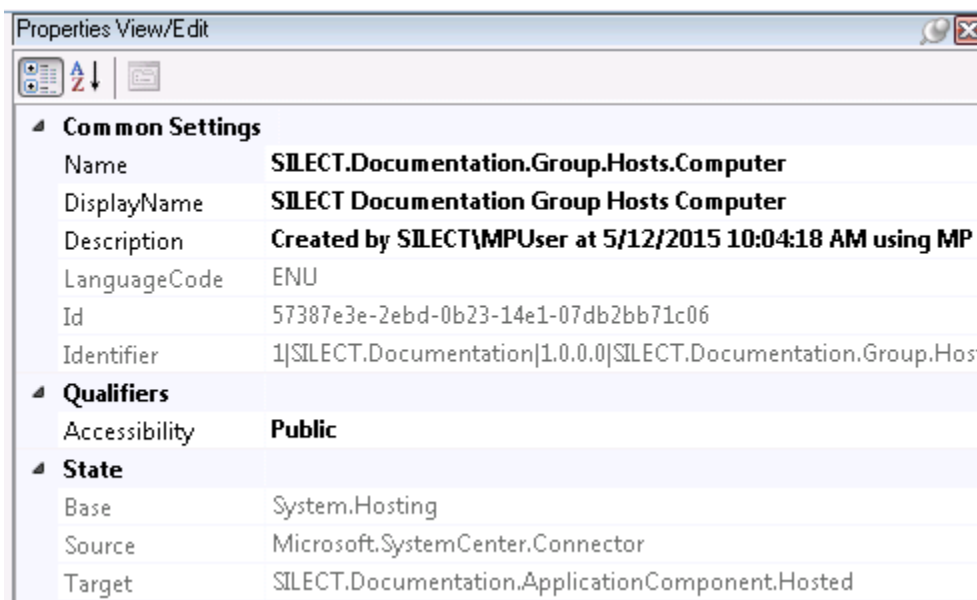
Relationships are used by Operations Manager to describe the relationship between objects or components. Relationships are also used to specify dependency rollup monitors. To add a new relationship to an MP, follow these steps:

- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 When the MP has been opened, right-click the **Relationships** container and select **Create New Relationship**.
- Step 3 Specify the type of the relationship. Note that **Hosting** relationships are often best created using the **Create New Class** wizard. Click **Next**.
- Step 4 Specify the source and target classes for the relationship. Click **Next**.
- Step 5 Enter the name, display name and description for the new relationship. Click **Next**.
- Step 6 Review the settings and click **Finish** to create the new relationship.

To save these changes, click the **Save** toolbar button or select **File, Save**.

To edit a relationship, follow these steps:

- Step 1 With the MP already opened, click the **Relationships** container to view the relationships currently defined in the MP.
- Step 2 Right-click one of the relationships displayed and select:
 - a. **Edit with Wizard...** to edit with the same wizard used to create the relationship,
 - b. **View/Edit XML** to edit the underlying XML,
 - c. **Delete...** to remove the relationships,
 - d. **Properties...** to view detailed properties of the relationship.
- Step 3 Or click on one of the relationships and change the properties which are not greyed out in the **Properties View/Edit** window.



To save changes, click the **Save** toolbar button or select **File, Save**.

Adding and Editing Secure References

(MP Studio & MP Author Pro only)

Secure references are used by Operations Manager to centralize and control the credentials used to perform various operations on target computers/devices. To add a new secure reference to an MP, follow these steps:

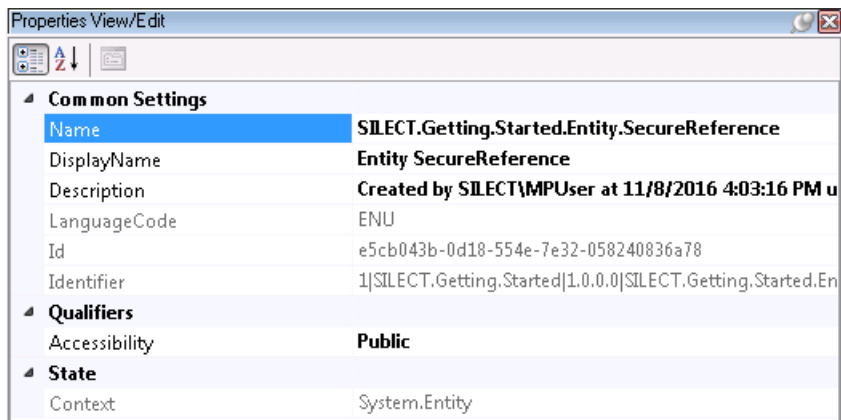
- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 When the MP has been opened, right-click the **Secure References** container and select **Create New Secure Reference**.
- Step 3 Specify the target class for the secure reference. This is typically **System.Entity (Object)**. Click **Next**.
- Step 4 Enter the name, display name and description for the new secure reference. Click **Next**.
- Step 5 Review the settings and click **Finish** to create the new secure reference.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Note The secure reference is a placeholder for credentials. Once the MP is imported into a management group, you will still need to use the Operations Manager console to specify the specific credentials to use.

To edit a secure reference, follow these steps:

- Step 1 With the MP already opened, click the **Secure References** container to view the secure references currently defined in the MP.
- Step 2 Right-click one of the secure references displayed and select:
 - a. **Edit with Wizard...** to edit with the same wizard used to create the secure reference,
 - b. **View/Edit XML** to edit the underlying XML,
 - c. **Delete...** to remove the secure references,
 - d. **Properties...** to view detailed properties of the secure reference.
- Step 3 Or click on one of the secure references and change the properties which are not greyed out in the **Properties View/Edit** window.



To save changes, click the **Save** toolbar button or select **File, Save**.

Adding and Editing Tasks

(MP Studio & MP Author Pro only)

Tasks are used by Operations Manager to perform diagnostic and remedial actions manually on operator command. Tasks have two different flavors: command and script. The command flavor executes a single command line program. The script flavor executes a script file of arbitrary complexity. To add a new task to an MP, follow these steps:

- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 When the MP has been opened, right-click the **Tasks** container and do one of the following:
 - a. select **Create New Windows Agent Command Task**.
 - b. select **Create New Windows Agent Script Task**.
- Step 3 Specify the target class for the task. Click **Next**.
- Step 4 Specify the action which is to be performed. Do one of the following:
 - a. Specify the command line to be executed, including the program and command line arguments. Target properties can be included in the arguments. Click **Next**.
 - b. Enter the name of the script or use the **Browse** button to find the script that will be used. You can optionally add parameters to the script. Click the **Add** button to add a

parameter to the script. Specify the name of the parameter, the initial value and the data type. Target properties can be included in the arguments. Repeat to include as many parameters as required. For PowerShell scripts, the list of parameters will be already filled in, but you will need to provide default values. Double click on the parameter and enter the parameter value as needed. Click **Next**.

Step 5 Enter the name, display name and description for the new task. Click **Next**.

Step 6 Review the settings and click **Finish** to create the new task.

To save these changes, click the **Save** toolbar button or select **File, Save**.

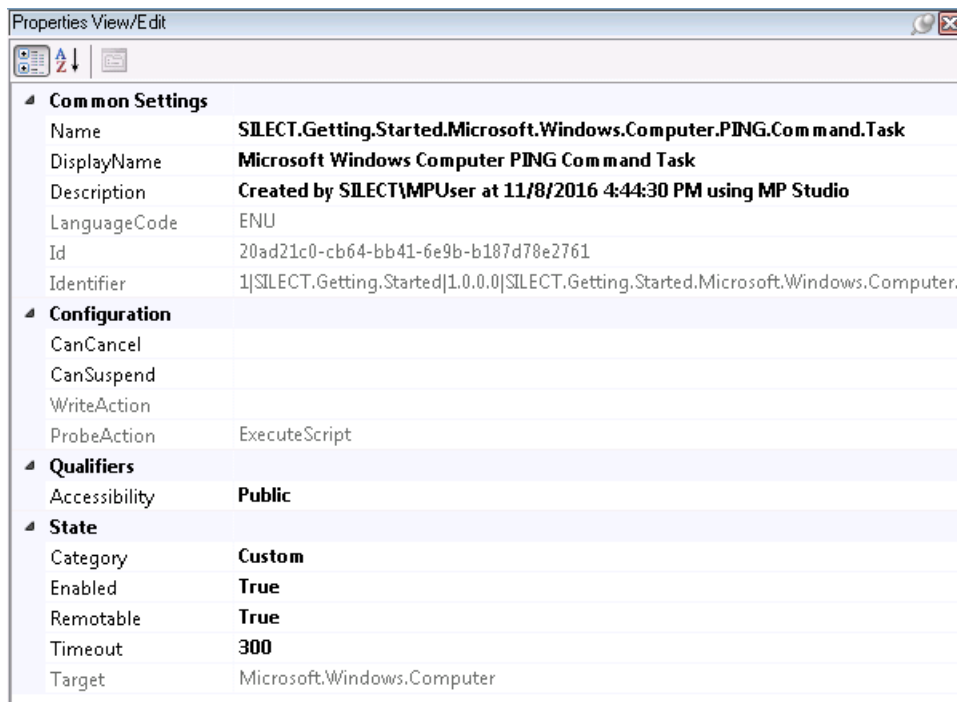
To edit a task, follow these steps:

Step 1 With the MP already opened, click the **Tasks** container to view the tasks currently defined in the MP.

Step 2 Right-click one of the tasks displayed and select:

- a. **Edit with Wizard...** to edit with the same wizard used to create the task,
- b. **View/Edit XML** to edit the underlying XML,
- c. **Delete...** to remove the tasks,
- d. **Properties...** to view detailed properties of the task.

Step 3 Or click on one of the tasks and change the properties which are not greyed out in the **Properties View/Edit** window.



To save changes, click the **Save** toolbar button or select **File, Save**.

Adding and Editing Diagnostics and Recoveries

(MP Studio & MP Author Pro only)

Diagnostics and recoveries are used by Operations Manager to automatically perform diagnostic and remedial actions. Each of these has two different flavors: command and script. The command flavors execute a single command line program. The script flavors execute a script file of arbitrary complexity.

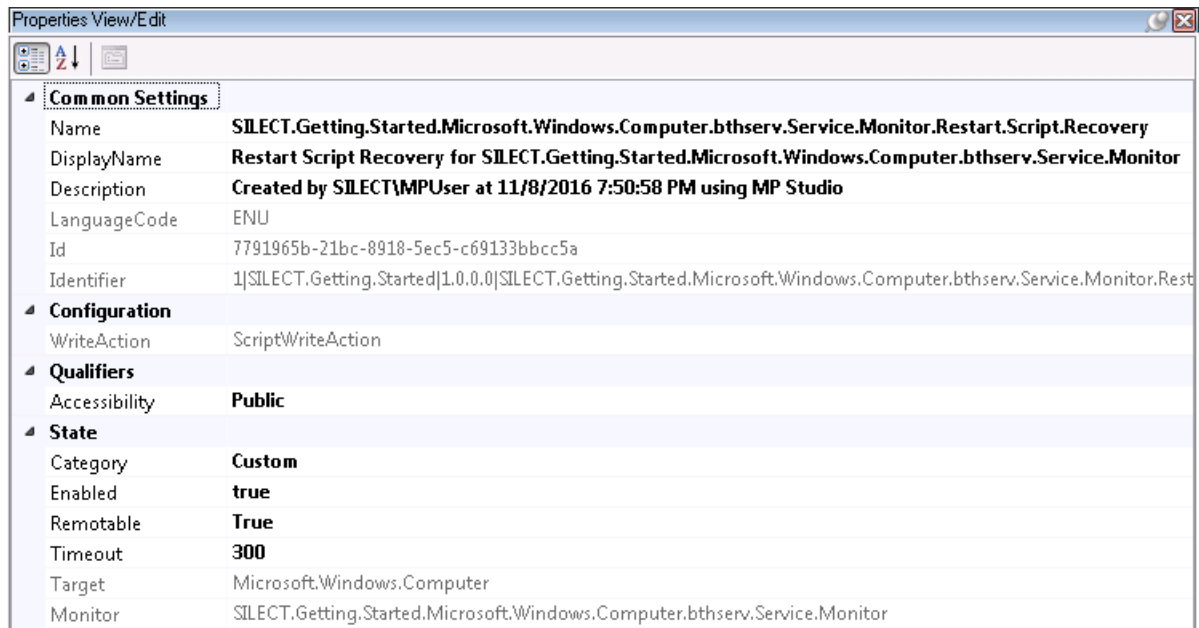
Diagnostics can be run automatically when a monitor enters a specified health state to provide additional diagnostic information. Recoveries can be run automatically when a monitor enters a specified health state to attempt to bring the system back into a healthy state. To add a new diagnostic or recovery to an MP, follow these steps:

- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 When the MP has been opened, do one of the following:
 - a. right-click the **Diagnostics** container and select **Create New Command Diagnostics**.
 - b. right-click the **Diagnostics** container and select **Create New Script Diagnostics**.
 - c. right-click the **Recoveries** container and select **Create New Command Recoveries**.
 - d. right-click the **Recoveries** container and select **Create New Script Recoveries**.
- Step 3 Specify the target class for the diagnostic or recovery. Click **Next**.
- Step 4 Specify the action which is to be performed. Do one of the following:
 - a. Specify the command line to be executed, including the program and command line arguments. Target properties can be included in the arguments. Click **Next**.
 - b. Enter the name of the script or use the **Browse** button to find the script that will be used. You can optionally add parameters to the script. Click the **Add** button to add a parameter to the script. Specify the name of the parameter, the initial value and the data type. Target properties can be included in the arguments. Repeat to include as many parameters as required. For PowerShell scripts, the list of parameters will be already filled in, but you will need to provide default values. Double click on the parameter and enter the parameter value as needed. Click **Next**.
- Step 5 Specify the health state which will automatically trigger the diagnostic or recovery. Click **Next**.
- Step 6 Select one or more monitors which you wish to add this diagnostic or recovery to. If more than one is selected, multiple diagnostics or recoveries will be created, one for each selected monitor. Click **Next**.
- Step 7 Enter suffixes for the name, display name and description for the new diagnostic or recovery. The name specified will be applied as a suffix to the name of the monitor to create the final name. Click **Next**.
- Step 8 Review the settings and click **Finish** to create the new diagnostic or recovery.

To save these changes, click the **Save** toolbar button or select **File, Save**.

To edit a diagnostic or recovery, follow these steps:

- Step 1 With the MP already opened, click the **Diagnostics** or **Recoveries** container to view the diagnostics or recoveries currently defined in the MP.
- Step 2 Right-click one of the diagnostics or recoveries displayed and select:
 - a. **Edit with Wizard...** to edit with the same wizard used to create the diagnostic or recovery,
 - b. **View/Edit XML** to edit the underlying XML,
 - c. **Delete...** to remove the diagnostics or recoveries,
 - d. **Properties...** to view detailed properties of the diagnostic or recovery.
- Step 3 Or click on one of the diagnostics or recoveries and change the properties which are not greyed out in the **Properties View/Edit** window.



To save changes, click the **Save** toolbar button or select **File, Save**.

Authoring with Fragments


(Full support in MP Studio & MP Author Pro. MP Author has a subset of fragments supported.)

Fragments allow an author to add elements to an MP, based on an XML template file. The fragment is loaded, and placeholders are replaced with actual values. Finally, the fragment is imported into the MP and all the elements from the fragment are merged into the existing MP.

Fragments can be produced locally, but most users will opt to start with the set produced by Kevin Holman, Kevin Justin, Silect and others from GitHub. The **Manage Fragments** dialog (**Tools, Manage Fragments...** menu item) contains a button to help you download the latest fragment library. Clicking the **Download fragment library from GitHub...** button will download the library from GitHub and unpack it into a local folder on disk. The GitHub library will be periodically updated, so checking for updates occasionally is recommended. Clicking the button will update the local set of fragments, without deleting any of your own (in the **My Fragments** sub-folder), so updating frequently is recommended.

If you do not have access to the internet, the download will fail, and a detailed message will be shown telling you how to download (on a system with internet access) and install the fragment library.

To add a new element from a fragment to an MP, follow these steps:

- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 When the MP has been opened, right-click the MP name at the top of the tree on the left-hand side of the console and select the **Import Fragment...** option; or use the toolbar button ( **Import Fragment**).
- Step 3 The **Import Management Pack Fragment** dialog will be displayed.
- Step 4 If you have not already downloaded the fragment library, click on the **Download fragment library from GitHub...** button at the bottom of the dialog. The library will be downloaded and installed (or updated) on disk.
- Step 5 Open a fragment file. Use one of the following methods.

- a. Use the **Browse...** button to locate one of the fragment (.mpx) files. The file names describe what the fragment will create. Click **Open**.
 - b. Use the **Search...** button to bring up the **Search for Fragments** dialog. This dialog allows you to filter the list of fragments down until you find the one you are interested in. You can filter by: element type created by the fragment; parameter names used in the fragment; keywords defined in the fragment; or a full text search. Select one of the fragment files (.mpx) and click **OK**.
- Step 6 In the **Parameters** area at the top, you will need to provide values for any parameter which is showing the error indicator (⊕). Many parameters will have **Browse...** buttons to allow you to select correct values from live systems. As each parameter value is entered, it will replace the placeholders in the **Preview** area. The current parameter will be **highlighted** in the **Preview** area to help make it clear where that value will be used. For convenience, parameter values are remembered, and parameters will be initialized with the last value when importing subsequent fragments.
- Step 7 Be sure to review the text in the Preview area in case there are things that will still need to be changed manually. Scripts may have items which need further editing. If edits are required, use the **Edit XML...** button at the bottom of the window to edit the XML (which will not have the parameter values replaced like you see in the **Preview** area).
-
- Note** The XML view/edit window allows control over whether the XML code is color coded and how attributes are wrapped. Right-click in the XML view/edit window and select **Colorize XML** and/or **Attribute Wrapping** to select your preference. These settings will be remembered.
-
- Step 8 When all the parameters have been specified, the **Import** button will be enabled. Click **Import** to add the elements defined by the fragment to the MP.

To save these changes, click the **Save** toolbar button or select **File, Save**.

Creating Fragments

(MP Studio & MP Author Pro only)

Creating your own fragments allows you to easily create customized workflows for your environment. Within MP Studio and MP Author Pro, users can select many existing elements (like rules, monitors, and discoveries) and create a fragment from the specified element and all required parts (module types, knowledge, etc.). The user controls which parts of the fragment use parameters to allow substitution during fragment import. The resultant fragment is saved to disk, where it can be used to create new elements, and can be shared with the fragment community on GitHub.

Note See [Appendix A: Fragments](#) for detailed technical information about fragments.

To create a new fragment from an MP, follow these steps:

- Step 1 Open a sealed or unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 When the MP has been opened, right-click any of the following elements and select **Create Fragment...**:
 - Monitors**
 - Rules**
 - Discoveries**
 - Overrides**
 - Tasks**

Diagnostics
Recoveries
Views
Folders
String Resources

- Step 3 In the **Create Management Pack Fragment** dialog, use the **Edit Header...** button to edit the description of the fragment and a version number (recommended). The last modified date will be automatically updated whenever the fragment is created or edited.
- Step 4 Add parameters by selecting some text and clicking the **Add Parameter From Selection...** button. Normally you will select either all the text between the start and end tag, or all or part of an attribute value, like the following:
`<ClassType ID="MPU.Spring2017.CNQ4807.Registry.Class"`
or
`Accessibility="Public"`
or
`<Path>SOFTWARE\Canon\WIA\Devices\CNQ4807\ProductId</Path>`
- If your selection looks like you have included part of the XML you will be asked to confirm your selection (to avoid inadvertently creating invalid XML).
- Step 5 In the **Add Fragment Parameter** dialog you will need to specify a name for the parameter. In some cases, a suitable name will be suggested, based on the surrounding XML tags. There are fly-out buttons on the right-hand side which will give you lists of common names that you might want to choose. The **Recent** button will show you a list of the names you have used recently. The **Browsable** button will show you a list of names which will allow you to browse for values when importing the fragment (very helpful!). The **All** button will show you a list of all parameter names used in all fragments on your system, to help you select from a common set of names.
- Step 6 After specifying a name, you can use the check boxes to control how the parameter is applied. Normally, the selected text will be replaced with the parameter name wherever it occurs in the fragment. By using the **Match case** and **Match whole** word options you can assure that replacement is only done where needed. The **Replace only this one occurrence** option only replaces the selected text, not everywhere the text occurs in the fragment.
- Step 7 Click OK and the parameter name is put in place of the selected text and in the header and list of parameters.
- Step 8 Repeat the above steps to add more parameters, if required.
- Step 9 You can add a parameter without selecting text (**Add Parameter...**), if required. You can also rename (**Rename Parameter...**) or delete (**Delete Parameter...**) parameters.
- Step 10 When the header has been edited and all the parameters have been defined, you may need to make some other edits to finalize the fragment. The **Edit XML...** button can be used to do that.

Note The XML view/edit window allows control over whether the XML code is color coded and how attributes are wrapped. Right-click in the XML view/edit window and select **Colorize XML** and/or **Attribute Wrapping** to select your preference. These settings will be remembered.

- Step 11 When finished, press the **Save As...** button to save the fragment. You will be prompted for a name for the fragment and allowed to specify which folder to save it in. Silect recommends you use the **My Fragments** folder to store your fragments. The **My Fragments** folder is treated specially and when updated fragments are loaded from

GitHub, fragment files in the **My Fragments** folder will not be touched (but will show up in the fragment search dialog).

The list of browsable parameter names lists only the base names. If you wish to have multiple browsable parameters (of the same type), you can add a prefix or suffix to the browsable name. The **Import Management Pack Fragment** dialog will still recognize them as browsable, and with different names the parameter values are handled as separate entities. For example, you may have a monitor which monitors one class and is hosted by a second class. To support this, you might name one parameter **MonitorClassID** and the second **HostClassID**.

There are several built-in parameters that can be manually added to a fragment (via XML editing) which may be useful. For instance, **FragmentVersion** will automatically be replaced with the version number from the fragment header, making it possible to include information about the fragments used into Description fields to make it easier to find outdated workflows when a fragment is updated. Please see Appendix A for more details about built-in and special rules about parameters.


If you have created a new fragment that you think might be useful for others, consider sharing it with the world on GitHub! Contact Silect support for details.

Editing Fragments

(MP Studio & MP Author Pro only)

Existing fragments can be edited, whether they are your own fragments, or fragments from the GitHub library. Editing is like creating fragments (above), so only the differences will be discussed here. The resultant fragment is saved to disk, where it can be used to create new elements, and can be shared with the fragment community on GitHub.

To edit a fragment, follow these steps:

- Step 1 Select the **Edit Fragment** button ( **Edit Fragment**) on the toolbar or select a fragment in the **Manage Fragments** dialog and click **Edit...** (see **Managing Fragments** below).
- Step 2 The **Edit Fragment** dialog will be displayed, showing the last selected fragment (if any).
- Step 3 You can use the **Search...** or **Browse...** buttons to select the fragment to edit (if not already loaded).
- Step 4 Edit the fragment as discussed above in **Creating Fragments**.
- Step 5 When finished, press the **Save** button to save the fragment using the existing file name. Or press the **Save As...** button to save the fragment with a new file name. You will be prompted for a name for the fragment file and allowed to specify which folder to save it in. Silect recommends you use the **My Fragments** folder to store your fragments. The **My Fragments** folder is treated specially and when updated fragments are loaded from GitHub, fragment files in the **My Fragments** folder will not be touched (but will show up in the fragment search dialog).
- Step 6 You can also **Browse** or **Search** to change the fragment being edited (although you will lose any changes).

Fragments can also have additional elements added to create more complex fragments. For example, a fragment might be created with a discovery (which will also include the class definition). Then a user could add a monitor to that fragment, as well as rules and/or views, etc. to create a fragment which defines the complete monitoring of certain types of application.

To add more elements to a fragment, follow these steps:

- Step 1 Open a sealed or unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 When the MP has been opened, right-click any of the following elements and select **Add to Fragment...**:
- Monitors**
 - Rules**
 - Discoveries**
 - Overrides**
 - Tasks**
 - Diagnostics**
 - Recoveries**
 - Views**
 - Folders**
 - String Resources**
- Step 3 You will be prompted to select an existing fragment file which the new elements will be added to.
- Step 4 Edit the fragment as discussed above in **Creating Fragments** and then save as discussed in Step 5 for **Editing Fragments** above.

If you have updated or created a new fragment that you think might be useful for others, consider sharing it with the world on GitHub! Contact Silect support for details.

Managing Fragments

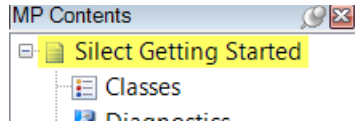
(MP Studio & MP Author Pro only)

To allow users to manage fragments, the **Tools, Manage Fragments...** menu item is provided. The **Manage Fragments** dialog allows the following operations.

- Users can change the folder where fragments are stored and where search will look for fragments. Use the **Browse...** button to select a different folder. By default, the users **Documents** folder is used, and fragments will be stored in the **Silect\Fragments** folder below **Documents**. Fragments will be stored in subfolders. Silect recommends customers keep their fragments in the **My Fragments** folder. Downloading fragments from GitHub will not touch fragments in the **My Fragments** folder, but will update, replace and delete fragments in the other sub-folders as needed.
- Users can download, or update, the library of fragments from GitHub using the **Download fragment library from GitHub...** button. Fragments in the **My Fragments** folder will not be touched by this operation.
- Users can select one or more fragments from the list of fragments or use the filter to find fragments of interest.
- When a single fragment is selected, the user may choose to edit (**Edit...** button), rename (**Rename...** button), or delete (**Delete** button) that fragment.
- When more than one fragment is selected, the user may choose to delete (**Delete** button) those fragments or combine (**Combine...** button) them into a single new fragment file.
- Clicking the **Combine...** button will merge the selected fragments into a new fragment file. The combined fragment will be loaded into the **Create Management Pack Fragment** dialog and further editing, and parameterization is as described above in **Creating Fragments**.

Management Pack Options

There are several options available for management packs that you work with within Silect MP Author. To access the available options, open an MP then right-click the MP name at the top of the tree structure in the left-hand side of the console.



The following MP options are available:

- View/Edit XML... ***
- Edit References...**
- Import Fragment...**
- Check For Missing Display Names...**
- Verify...**
- Run Best Practices Analyzer...**
- Seal...**
- Deploy...**
- Compare ***
- Document ***
- Test... ***
- View Reference Tree ***
- View Summary ***
- Show All Elements**

* Feature available only in Silect MP Studio or Silect MP Author Professional.

The following sections describe how to use each of these features.

View/Edit XML (MP Studio & MP Author Pro only)

To view or edit the XML code for an entire MP, follow these steps:

- Step 1 Open an MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 Right-click the MP name at the top of the tree on the left-hand side of the console.
- Step 3 Select the **View/Edit XML...** or **View XML...** option.
- Step 4 A new **Edit – 'MP Name'** window will be shown. If the file is unsealed, the XML code can be edited. Sealed MPs will be displayed but the XML cannot be edited.
- Step 5 Make any required changes to the XML and click **OK**.

Note The XML view/edit window allows control over whether the XML code is color coded and how attributes are wrapped. Right-click in the XML view/edit window and select **Colorize XML** and/or **Attribute Wrapping** to select your preference. These settings will be remembered.

CAUTION! Detailed knowledge of the MP XML schema is required to successfully make changes to the XML form of an MP.

Edit References

MPs reference information and libraries in other MPs. To add a reference to an MP, follow these steps:

- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 Right-click the MP name at the top of the tree on the left-hand side of the console.
- Step 3 Select the **Edit References...** option.
- Step 4 Click the **Add / Update Version...** button.
- Step 5 Click one or more MPs (use the Ctrl click option to multi-select) the click **OK** when done.
- Step 6 Click **Close**.

Note Selecting an existing reference MP, will update the reference to the version of the newly selected MP. This allows an easy way to update references as new MPs are released.

The new references will be added to the MP. To save these changes, click the **Save** toolbar button or select **File, Save**.

To remove a reference from an MP, follow these steps:

- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 Right-click the MP name at the top of the tree on the left-hand side of the console.
- Step 3 Select the **Edit References...** option.
- Step 4 Select one or more references from the list.
- Step 5 Click the **Remove** button.
- Step 6 Click **Close**.

The references will be removed from the MP (if no elements are using these references). To save these changes, click the **Save** toolbar button or select **File, Save**.

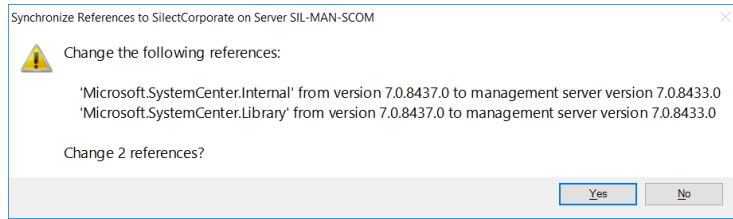
To change the alias of a reference in an MP follow these steps:

- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 Right-click the MP name at the top of the tree on the left-hand side of the console.
- Step 3 Select the **Edit References...** option.
- Step 4 Select one references from the list.
- Step 5 Click the **Edit Alias...** button.
- Step 6 Enter the new alias in the **Edit Reference** dialog and press **OK**.
- Step 7 Click **Close**.

The references alias will be renamed in the MP. To save these changes, click the **Save** toolbar button or select **File, Save**.

To ensure the reference versions in an MP are in sync with your management group follow these steps:
(MP Studio only)

- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 Right-click the MP name at the top of the tree on the left-hand side of the console.
- Step 3 Select the **Edit References...** option.
- Step 4 Click the **Synchronize with Management Server...** button.
- Step 5 Enter the name of a management server and the credentials as needed.
- Step 6 If there are different versions of the referenced MPs on the management server, then a dialog like the following will be displayed.



- Step 7 If you want to update the reference version numbers, click **Yes**. Otherwise, click **No**.
- Step 8 Click **Close**.

The references versions will be updated in the MP. To save these changes, click the **Save** toolbar button or select **File, Save**.

Import Fragment

See [Authoring With Fragments](#) above for complete details.

Check For Missing Display Names

Management pack elements all have a display name which is shown in the Operations Manager console. If the display name is missing, the identifier is usually shown, but these cannot be localized. Display names should be provided to give the best user experience. To check for missing display names, follow these steps:

- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 Right-click the MP name at the top of the tree on the left-hand side of the console.
- Step 3 Select the **Check For Missing Display Names** option.
- Step 4 A list of elements and sub-elements with missing display names will be shown. Select each you wish to edit and click **Edit**, or you can double-click the item. You can add missing **Display Names** and **Descriptions**. Click **Save** when done.
- Step 5 After each item has the missing display names defined, it will be removed from the list making it easy to keep track of your progress.
- Step 6 When finished, click **Close**.

The new display names and descriptions will be added to the MP. To save these changes, click the **Save** toolbar button or select **File, Save**.

Note Elements created with SCOM templates (Database monitor and Web Site monitor) will have many missing display names as the SCOM template does not define display names in all cases.

Verify

MPs created or edited within Silect MP Author can be verified before being imported into Operations Manager. To verify an MP, follow these steps:

- Step 1 Open an MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 Right-click the MP name at the top of the tree on the left-hand side of the console.
- Step 3 Select the **Verify** option.

The MP will be verified, and any problems or issues will be identified, and a detailed message will be provided.

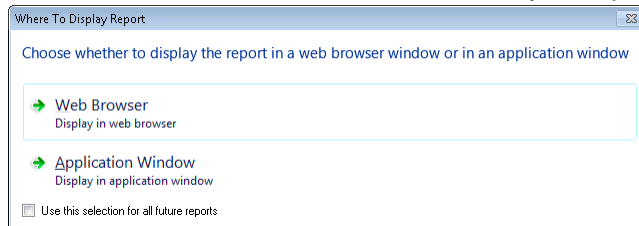
Note The Verify command will warn about Classes and Relationships in unsealed MPs (but will not

give warnings once the MP is sealed). These warnings can usually be ignored.

Run Best Practices Analyzer

MPs created or edited within Silect MP Author can be run through the Microsoft MP Best Practices Analyzer (MPBPA) before being imported into Operations Manager. To run the MPBPA on an MP follow these steps:

- Step 1 Open an MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 Right-click the MP name at the top of the tree on the left-hand side of the console.
- Step 3 Select the **Run Best Practices Analyzer** option.
- Step 4 You will be asked to choose the location for your report, select your preferred option.



The MP Best Practices Analyzer will be run, and the results will be displayed.

-
- Note** MPBPA is provided as part of this software but is written and maintained by Microsoft. MPBPA has some issues that should be noted.
- MPBPA requires an exact version match for referenced MPs where MP Author and MP Studio will allow a match of a reference MP with the same or greater version number. You may find that MPBPA does not work properly unless you acquire the exact versions referenced.
 - MPBPA does not accept Management Pack Bundle (.mpb) files. If your MP references MPB files, then MPBPA will not be able to validate your MP file.
 - MPBPA uses .NET 2.0/3.5. You may need to enable/install the earlier version of .NET to allow MPBPA to run.
-

Seal

MPs created or edited within Silect MP Author can be sealed before being imported into Operations Manager. Sealing can optionally save the sealed MP to the Store (if available) and can optionally deploy the newly sealed MP to a management server. To seal an MP, follow these steps:

- Step 1 Open an unsealed MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 Right-click the MP name at the top of the tree on the left-hand side of the console.
- Step 3 Select the **Seal** option.
- Step 4 Provide the name of the folder where the sealed MP will be saved, the name of the Key File as well as the Company name. An optional copyright string may be specified.
- Step 5 If the Store database is available, check the **Add to Store?** check box to have the newly sealed MP saved to the Store database.
- Step 6 If you wish to deploy the newly sealed MP, check the **Deploy?** check box and select a management server from the drop-down list.
- Step 7 Click **OK**.

The MP will be sealed using the information provided. If you choose the default values, then the MP will be sealed using a Silect MP Author specific license, which should not be considered secure enough to use for anything other than testing purposes. The newly sealed MP will also be added to the Store database (if requested) and/or deployed to a management group. All settings will be remembered and used to initialize this dialog the next time this operation is performed.

Deploy

MPs created or edited within Silect MP Author can be deployed directly to Operations Manager. To deploy an MP, follow these steps:

- Step 1 Open an MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 Right-click the MP name at the top of the tree on the left-hand side of the console.
- Step 3 Select the **Deploy** option.
- Step 4 Specify the name of the Operations Manager server to connect to. If different credentials are required enter them in this dialog as well.
- Step 5 Click **OK**.
- Step 6 The Deploy Management Pack dialog will be displayed showing which MPs and their versions are to be deployed. If reference MPs need to be deployed (because they are missing or out of date on the management server), they will be listed here as well. Click **Deploy**.

The MP will be deployed to Operations Manager as a background task. A message box will be shown when the operation is complete.

Compare (MP Studio only)

MPs can be compared to other MPs to determine the differences. To compare an MP, follow these steps:

- Step 1 Open an MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 Right-click the MP name at the top of the tree on the left-hand side of the console.
- Step 3 Select the **Compare** option, followed by selecting where the second MP is located.
- Step 4 Use the browser to identify the second MP. Click **OK**.

A new window will open showing the differences between the two MPs.

Document (MP Studio only)

MPs can be documented by producing an HTML report or an Excel spreadsheet. To document an MP, follow these steps:

- Step 1 Open an MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 Right-click the MP name at the top of the tree on the left-hand side of the console.
- Step 3 Select the **Document** option, followed by selecting the type of document to create.
- Step 4 For Excel, use the browser to specify where to store the Excel document. Click **OK**.

A new window will open documenting the MP. You can also choose to schedule documenting to Excel workbooks.

Test (MP Studio only)

MPs can be tested before deployment to determine what to expect in production. This is an advanced feature. See the MP Studio documentation for details.

View Reference Tree (MP Studio only)

MPs reference other MPs to provide classes, relationships, monitor types, etc. To view the reference tree of an MP, follow these steps:

- Step 1 Open an MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 Right-click the MP name at the top of the tree on the left-hand side of the console.
- Step 3 Select the **View Reference Tree** option.

A new window will open showing the reference tree for the MP. Click on each MP to see the relationships for that MP. Right-click for more options.

View Summary (MP Studio only)

MPs can contain numerous elements such as rules, monitors, discoveries, etc. To view the summary of an MP, follow these steps:

- Step 1 Open an MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 Right-click the MP name at the top of the tree on the left-hand side of the console.
- Step 3 Select the **View Summary** option.

A new window will open showing a summary for the MP.

Show All Elements

MPs can contain many elements such as rules, monitors, discoveries, etc. Normally only the elements which are present, or can be created, will be shown. To view all elements of an MP, follow these steps:

- Step 1 Open an MP by clicking **Open** or click **File, Open, From File** or click **File, Recent** to open a recently opened MP.
- Step 2 Right-click the MP name at the top of the tree on the left-hand side of the console.
- Step 3 Select the **Show All Elements** option.

The list of MP elements will be redrawn showing all elements. Repeat this procedure to show the more limited set of elements.

Chapter 4

Getting Technical Support

Silect MP Author users have several options for technical support.

Self–Service Support Portal

Users can access FAQs, training videos and other Silect MP Author support information at:

<https://www.silect.com/mp-author-support>

Electronic Support

A ticketed support system is also available for registered MP Author Pro and MP Studio users who cannot find the information required to help them address their issue on the Silect MP Author support portal (<https://www.silect.com/mp-author-support>). Registered users can email support@silect.com with the following information:

- A detailed description about the problem
- The Silect MP Author log files (located via the **View – MP Author Log Folder** menu item)
- Screen shots if applicable
- The management pack you are having problems with.

Response times are not guaranteed using this support service for MP Author customers. MP Studio and MP Author Professional customers will get support priority.

Including the log files corresponding to the time that the problem occurred **will** improve the ability of support to help with your problem.

Premium Support

Silect Software also makes a pay-based premium support service available. This service provides telephone and electronic support with guaranteed response times. For more information on Premium Support please go to <https://www.silect.com/mp-author-support>

Other Resources

<https://www.youtube.com/user/SilectSoftware> is a YouTube video channel published by Silect, that contains useful videos describing how to use Silect products, such as MP Author.

Appendix A: Fragments

Built-In Parameters

The following table lists the built-in parameters available to every fragment. Users will not need to fill in values for these parameters. The values will automatically be filled in when the fragment is loaded, prior to importing it into the MP. To use these parameters, fragment authors will need to edit the XML of their fragment and add them manually. These would often be used in **<Description>** fields to allow searching for those values with external tools.

Name	Sample Value	Notes
Date	15-Oct-18	The value of the ##Date## parameter will be the current date using the system short date format.
DateTime	15-Oct-18 13:30:05	The value of the ##DateTime## parameter will be the current date and time using the system short date format and long time format.
FragmentName	MyFragment.mpx	The value of the ##FragmentName## parameter will be the name of the fragment file.
FragmentVersion	1.0	The value of the ##FragmentVersion## parameter will be the version number specified in the fragment header (Version: field).
FragmentLastModified	12-Oct-18	The value of the ## FragmentLastModified## parameter will be the last modified date specified in the fragment header (LastModified: field), which is automatically updated whenever the fragment is saved.

These built-in parameters can be useful for recording dates and versions of fragments in the MP, so that when a fragment is updated it is possible to find all MPs which used the old version of the fragment and make suitable updates.

Browsable Parameters

The following table lists the names of parameters which will display the **Browse...** button allowing users to browse for information, instead of typing in values, which can be error prone. The names specified in the following table can have prefixes or suffixes added, but not both, and will still be recognized. For example, this allows multiple classes to be browsed, by having a parameter named **ClassIDSource** and a second parameter named **ClassIDTarget**. The will be treated as two separate parameters, but both will show the **Browse...** button.

Name	Browser Type
ClassID	Class browser

CounterName	Performance counter browser
EventID	Event log browser
EventSource	Event log browser
FrequencySeconds	Time interval browser
InstanceName	Performance counter browser
IntervalSeconds	Time interval browser
LogFileDirectory	File system folder browser
LogName	Event log browser
ObjectName	Performance counter browser
OID	SNMP variable browser
ProcessName	Process browser
RegKeyPath	Registry key browser
RegValuePath	Registry value browser
ServiceDisplayName	Windows NT service browser
ServiceName	Windows NT service browser
TimeoutSeconds	Time interval browser
WMIQuery	WMI database browser

Parameters which are related (like ServiceName and ServiceDisplayName) will have both values filled in with a single browse. This also applies for event logs and performance counters.

Special Parameter Rules

There are some other rules about parameters and parameter values that you may want to be aware of.

Sanitizing

Normally when parameters are replaced in SCOM names (for instance ID, Target, etc.) the parameter values will automatically be sanitized. SCOM has a restrictive set of rules regarding what characters can be in a name (for instance, no spaces), so when MP Author puts a parameter value into a name, it will automatically remove or replace special characters, so the result is a legal SCOM name. In places where then parameter value is not used in a name (for instance, a **<Description>**), the value is not modified. An example of this is the names of Windows NT services. Typically, these are a single word, like **BITS**, but there are services with multiple word names, like **HP LaserJet Service**, which need to be sanitized.

There are cases where a parameter value is being used in a place which does not appear to be a name, but the value needs to be sanitized. In those cases, the parameter name can have the suffix **SanitizeForUseInElementID** added to force the value to be sanitized. For instance, **##ServiceNameSanitizeForUseInElementID##**.

Sanitized values have some special rules. If a parameter name ends with **Path**, or the name is **RegistryKey** or **RegistryValue**, the sanitized version of the value is just the last portion of the value after the last \ (if any).

Sanitized values may consist of letters, numbers, periods (.) or underscores (_). All other characters are removed or replaced. The slash character (/) is replaced with **Per**. The percent character (%) is replaced with **Percent**. Space characters are replaced with underscores (_). Multiple consecutive underscores or periods are replaced with single characters.

Required References and Aliases

Parameter values may be replaced in the **RequiredReference**: sections of the header. Aliases will be updated in parameter values, if possible.