

Altair® Monarch® v2021.1.1
MONARCH SERVER
ADMINISTRATOR'S GUIDE
Automator | Report Mining Edition

TABLE OF CONTENTS

Introduction	1
About Monarch Server Automation Edition	1
Monarch Server Automation Edition Workflow	2
About Monarch Server Report Mining Edition	4
Launching Monarch Server Web Administrator	4
Navigation Tree	7
Functional Buttons	9
Page Navigation	9
Using the Calendar	10
Report Warehouse	11
Maintenance Management	11
Log Clearance Tasks	11
Maintenance Processes	13
Automator	18
Server Status	18
Standard Processes	20
Processes	20
Schedules	38
Jobs	39
Distributions	41
Alerts	49
Global Environment Scripts	55
Project Editor	55
Visual Processes	74
Processes	75
Schedules	205
Process Templates	205
Process Logs	206
Alerts	207
Global Environment Scripts	210
Script Editor	210
PreExport Script	211
PostExport Script	211
PreProcess Script	212
PostProcess Script	212
Global Declarations Script for Standard Processes	212
Global Declarations Script for Visual Processes	213
References and Imports	213

Passwords	213
Inserting a Script for Standard Processes.....	213
Inserting a Script for Visual Processes	216
Testing a Script.....	217
Connection Definitions	218
Using the Naming Macros.....	219
Offset Expressions in Macros.....	223
When a Job Fails to Run.....	224
Runtime Fields	224
Project Inputs via Manifest	226
Project Outputs via Manifest	227
Standard Processes with Manifest Monitoring	230
Visual Processes with Manifest Monitoring	235
XSD for Standard Processes	240
XSD for Visual Processes	242
RMS	253
Content Recognition Rules	253
Creating or Editing Content Recognition Rules.....	253
Portlets.....	255
Portlets.....	255
Adding a Portlet	255
Editing a Portlet	256
Deleting a Portlet	257
Portlet Groups	257
Adding a Portlet Group	257
Editing a Portlet Group.....	258
Deleting a Portlet Group	258
Server Library.....	259
Folders	259
Adding a Folder	259
Editing a Folder.....	261
Deleting a Folder	262
Projects	262
Adding a Project File.....	263
Editing a Project File	263
Opening a File in the Project Editor	265
Deleting a Project File.....	266
Changing a Project File State	266
Models	266
Adding a Model.....	267
Editing a Model	267

Deleting a Model	270
Changing Model State	270
Templates	270
Adding a Template.....	271
Editing a Template	271
Deleting a Template.....	273
Changing Template State	274
Workspaces	274
Adding a Workspace.....	274
Editing a Workspace	276
Deleting a Workspace.....	278
Changing Workspace State	278
Change Lists	279
Adding a Change List.....	279
Editing a Change List.....	280
Deleting a Change List.....	282
Changing Change List State	282
Bulk Edits	282
Rights and Privileges.....	285
Assigning Privileges	285
Automator Administrator	286
Automator User.....	286
Automator Light User	286
Assigning Access Rights.....	287
Setting Security Provider.....	287
Roles.....	288
Adding a Role	289
Editing a Role	290
Deleting a Role	291
User Groups.....	291
Adding a User Group	291
Importing a User Group	293
Editing a User Group	293
Deleting a User Group	294
Users.....	294
Adding a User	294
Changing User Password	296
Editing a User	296
Deleting a User	297
System Settings	298
System Preferences.....	298
Application Settings	300
About Information	301

System Defaults.....	302
Time Intervals	302
New User Defaults.....	303
Global User Settings.....	305
PDF Default Print Settings.....	305
Automator Settings	306
Locations Management.....	314
Named Paths	315
RMS Settings.....	317
Exception Dates.....	319
Locale Management.....	320
Bulk Schedules Edits	321

News..... 323

All News	323
Expired News	323
Global News.....	324
Adding a News Message	325
Editing a News Message.....	325
Deleting a News Message	326

System Reports..... 327

Configuration Reports	327
Product License Report	327
Roles Report.....	328
User Groups Report.....	328
Users Report.....	329
Security Configuration Report.....	329
Automator Process Dependencies	330
Transaction Reports.....	330
Security Report	331
System Report.....	332
Added Users Report	332
Deleted/Suspended Users Report	333
Users Not Logged In Report	333
Dynamic Login Requests Report	334
Audit Entity Actions Report	335
Custodian Schedules Logs Report	335
Automator Distribution Audit	336
Converting a Report to Another Format	337
Active Users.....	337

Technical Support..... 338

Glossary..... 340

INTRODUCTION

Welcome to Monarch Server (MS), Altair Engineering, Inc.'s latest reporting solution.

Monarch Server is the smart business intelligence software that gives you the power to keep the data and lose the paper. That's because Monarch Server's powerful report mining technology can unlock data buried in reports and documents and turn it into data that you can use now.

Monarch Server comprises three modules, namely, Automator, Report Warehouse, and RMS. This guide provides a comprehensive look at the following:

- ❑ Automator

Automator is designed to be used in conjunction with Altair Monarch Complete, a data extraction and analysis tool that lets you view, analyze, extract, and print data from existing computer reports. Any structured report that is used in your organization can be utilized by Monarch Complete. When Monarch Complete "reads" a report file, instead of producing a hard copy printout with words and numbers frozen on the page, Monarch Complete creates a soft copy of the report on screen, with live data you can work with.

- ❑ Report Mining Server

Report Mining Server is a powerful tool that allows you to extract data from structured documents or reports. Altair has further developed Monarch Server Report Mining Edition with the ability to provide the extracted data in various analytical and presentation formats for delivery over the Web. Monarch Server Report Mining Edition employs innovative technology that allows any PC or laptop connected to the Internet, or to a corporate intranet, to access reports stored in other systems. All of these capabilities make Monarch Server Report Mining Edition truly unique.

The topics in this section provide basic information about the following:

- ❑ [Monarch Server Automation Edition](#)
- ❑ [Automation Edition Workflow](#)
- ❑ [Monarch Server Report Mining Edition](#)
- ❑ [Launching MSAdmin](#)
- ❑ [Navigation Tree](#)
- ❑ [Functional buttons](#)
- ❑ [Page Navigation](#)
- ❑ [Using the Calendar](#)

ABOUT MONARCH SERVER AUTOMATION EDITION

Monarch Server Automation Edition is a data conversion solution that uses existing ASCII or ANSI reports, PDF files, XPS files, ISAM files (for example, XLS, MDB, ACCDB, DBF) Open XML Spreadsheets (XLSX/XLSM), HTML files, and OLEDB/ODBC sources as a source for data. You can input report files (for example, TXT, DAT, PRN, LOG) into Monarch Server Automation Edition and then export their data to a variety of formats (for example, PDF, XLS, XLSX/XLSM, MDB, ACCDB, HTML, DBF) or to an OLE DB or ODBC destination. Monarch Server Automation Edition also allows you to export files to a Microsoft Office SharePoint Server, FTP site, email destination, or distribution list.

In short, Monarch Server Automation Edition lets you extract, transform, move and load data faster and more easily than previously possible.

Monarch Server Automation Edition offers far more than an easy way to automate Monarch: it is an information delivery system, capable of distributing files over the network and to SharePoint locations, emailing users the files they need, or alerting them that new files are available and where to get them, either over the network or via the Internet. RSS feeds can be created, integrating with newsreaders or corporate information portals, so users are kept up-to-date on the latest information and can click to download or view the information via their browser.

Monarch Server Automation Edition is a perfect solution for extracting data from periodic reports on a regular basis. For example, you might instruct Monarch Server Automation Edition to process files delivered to a specified location on your network at 3:00AM, delivering Excel files to all sales reps so that they have the latest figures in their Inbox first thing in the morning.

Alternatively, you can use Monarch Server Automation Edition to monitor specified folders, so that as soon as a file arrives, the conversion process begins.

Monarch Server Automation Edition is also great for automating complex conversion projects that call for extracting data from multiple reports, generating output for multiple recipients, or for taking multiple passes through a complex report in order to assemble the data you need.

Features and Functions

Monarch Server Automation Edition has the following capabilities:

- Automated conversion of data from TXT, PDF, ISAM, XPS, and OLE DB/ODBC sources to an ISAM, or OLE DB/ODBC (for example, SQL Server, Oracle) target by scheduled or event-driven execution via monitored folders.
- Email distribution of PDF, ISAM, and Open XML Spreadsheets
- File system and SharePoint distribution of input and export files
- RSS publication
- Creating logs and alerts resulting from operations
- Allowing user creation of pre-processes and post-processes using VB.NET Scripting
- Scheduled administration of job logs and database entries

MONARCH SERVER AUTOMATION EDITION WORKFLOW

When using Monarch Server Automation Edition a typical sequence of events might be as follows:

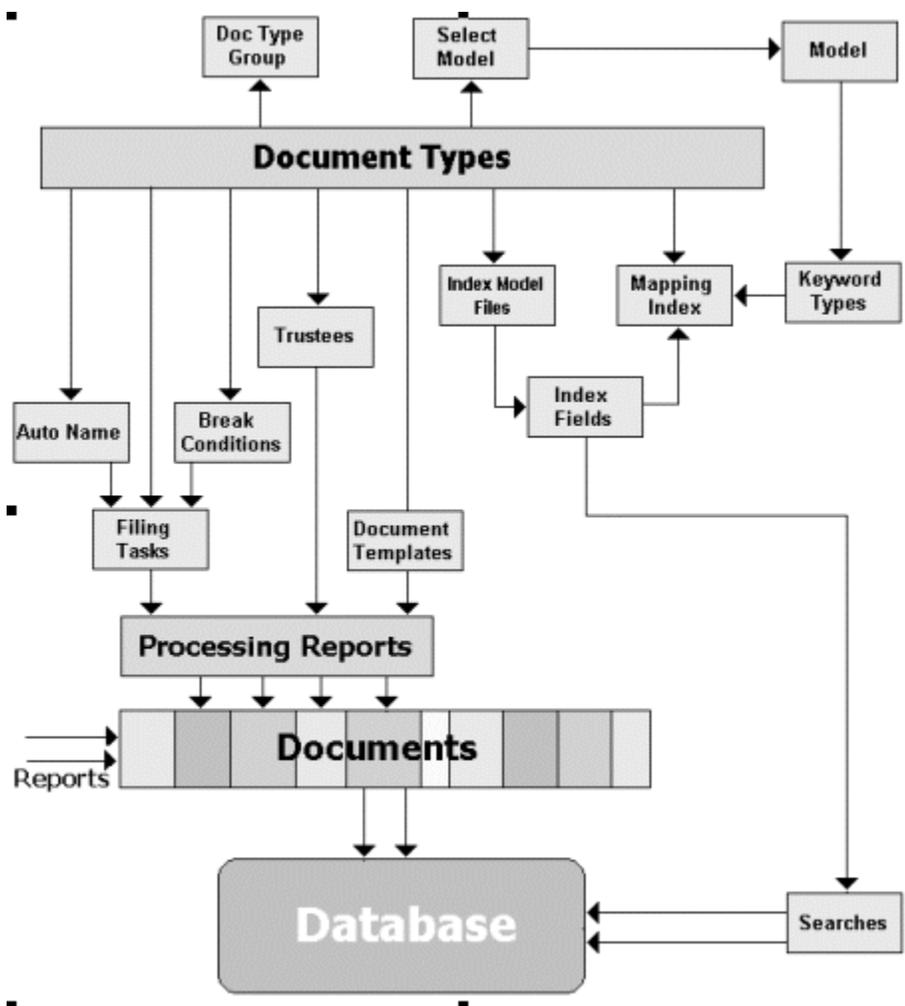
NOTE This sequence of events does not have to be followed exactly. Depending on your needs, you may perform fewer or more steps than those listed below, and the order in which you perform them may differ.

1. [Create a Process](#): Once you have created or identified the project file(s) you want to work with, you can create a process. Creating a process includes multiple steps, such as the following:
 - **Specify general settings**: General settings include a name and description of the process, a How many projects per job setting (i.e., one per job or multiple ones per job), the project(s) to be assigned to the process, etc.
 - **Specify advanced settings**: Advanced settings include a priority setting, and whether to allow parallel jobs and process retries.

- **Specify job log disposition settings:** These settings determine what will be done with the job log after the job has run, and when (i.e., under what circumstances) the specified action will be carried out.
- **Specify input file monitoring settings:** These settings enable you to tell Monarch Server Automation Edition to monitor a particular location, then trigger the running of a process when a specific input file (or any file of the specified file type) appears in that location.
- **Specify script settings:** Script settings enable you to specify pre-process and post-process operations that execute before or after any operations in the process itself do.

NOTE Depending on your needs, not all of these steps will have to be performed, for example, you may not need to specify input file monitoring settings.

2. [Run the Process:](#) Once you have completed the process configuration, you can either run the process manually, schedule the process to run, or configure an input file specification which will trigger the process to run when a particular input file arrives in the specified location.



Here the [Document Type](#) is a base concept, which defines how the system functions as a whole. You may acquaint yourself with it further by reading the section titled [Document Types](#). Document types comprise a [Document Type Group](#), as described in the section titled [Document Groups](#).

When you create or edit a document type, the system allows you:

- ❑ To select a model from the list of available models. Before you can select a model, you need to define it with the help of the Model page, described in the section titled [Adding a Model](#). Each model described by [mod- or xmod-type files](#) defines the [keyword types](#).
- ❑ To define the auto-name scheme using the AutoName page, which automatically assigns document names based on any combination of indexes and document identifiers. For more information, see [Auto-naming documents](#).
- ❑ To define [filing tasks](#), which begin to process the reports. For information on how to manage filing tasks see [Filing Tasks](#).
- ❑ To set trustees ([users](#)) for documents in the system defined by their [roles](#) and the [user group](#) they belong to. For details refer to [Rights and Privileges](#).
- ❑ To define [document templates](#) for documents extracted from the reports. On how to define document templates, see [Templates](#).
- ❑ To select the [index model files](#) with the help of the Model page described in the section titled [Adding a Model](#). These files define the [index fields](#) used to search for documents in the database.
- ❑ To bring [index fields](#) in accordance with [keyword types](#) with the help of the Document Type page. For more information, see [Adding a Document Type](#).

ABOUT MONARCH SERVER REPORT MINING EDITION

The RMSAdmin is a part of the Monarch Server Report Mining Edition system, which consists of these major components:

- ❑ **SQL database**
Monarch Server Report Mining Edition works with industry-standard SQL databases. It supports Microsoft SQL Server 2008+. A database server can be chosen from a list of supported devices based on your report volume and existing network architecture. This database supplies the underlying mechanisms for managing the internal data structures.
- ❑ **RMS Administrator (RMSAdmin)**
It is used by the system administrator to define the parameters of the system. RMSAdmin provides capabilities for adding models, enforcing system security, establishing user rights and privileges.
- ❑ **RMS Client (RMSCClient)**
It contains the functionality necessary for viewing and analysis of the data from reports.

LAUNCHING MONARCH SERVER WEB ADMINISTRATOR

MSAdmin is the administration interface of Monarch Server that operates in an Intranet/Internet environment. Hence, users should be familiar with the standard techniques used to work in the Internet environment.

NOTE Only users with administrator privileges have access to the system.

To initiate MSAdmin

1. Launch **Microsoft Internet Explorer** or any other browser by clicking its shortcut.
2. Type the MSAdmin address in the **Address** bar, which may look like this: <http://servername/MSAdmin>.

NOTE In Monarch Server that is upgraded from previous versions (e.g., v13.2), the virtual folders could still be labeled as "DSAdmin" (or "DSClient"). In this case, use "DSAdmin" (or "DSClient") instead of "MSAdmin" (or "MSClient") to open the Admin (or Client) page.

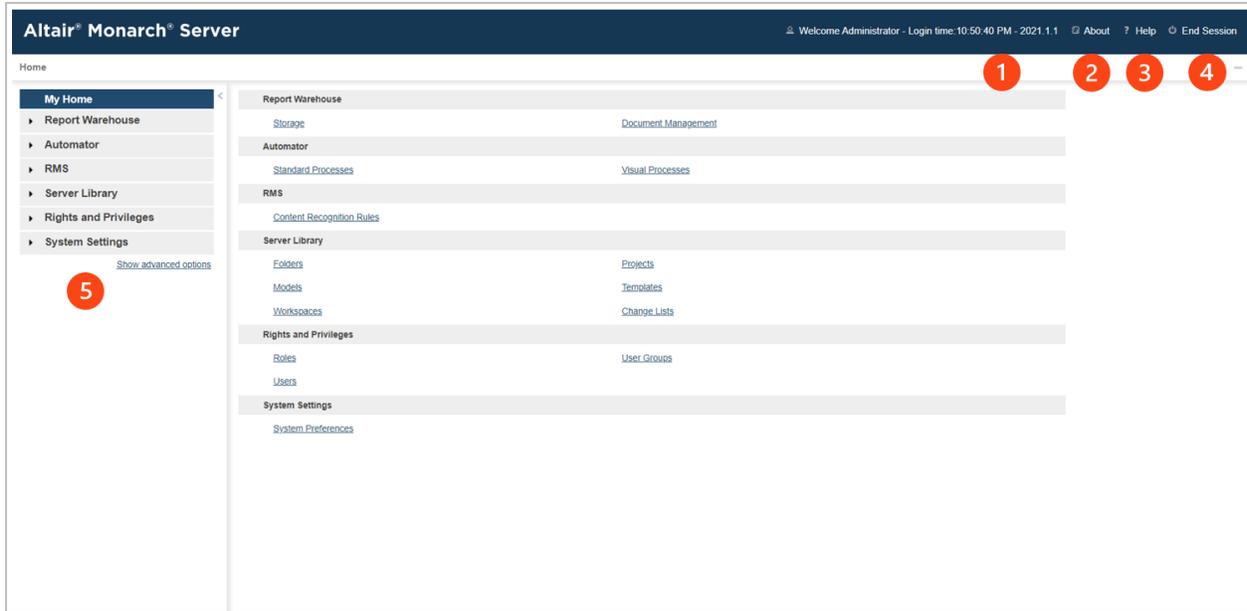
3. Press **Enter**. Your browser will load the Login page.
4. Select a locale in the upper right corner of the page. The list includes locales that were selected during the installation.
5. Type in your name as an administrator, the password, and click **Sign In**. If you enter your password correctly, the main page of Monarch Server Administrator is displayed on your screen.

NOTE If your password has expired, the system displays the expired password page. You have to change your password. Type in your old password, a new password, confirm the new password and then click **Save**. If the new password you entered does not correspond to the requirements (for example, the length of a new password is less than the value defined by the administrator), the system displays a corresponding message.

If you try to log in to the application again while running it, a message will appear on the Login page, informing you that you have already logged in to the application. To clear the current session and log in to a new one, select the check box below the message, and click **Sign In**. This option is available only for the **Named** license type.

Main Page of MSAdmin

Upon logging in successfully, the main page of MSAdmin is displayed.



Panels and Functions	Description
1	<p>Login Details</p> <p>This shows the last login time and the current version of the application.</p>
2	<p>About</p> <p>Clicking this button opens the About window that shows details such as product version, assembly version, SDK version, and DataPrep version. Company information and a link to the list of open-source software used are also shown here.</p> <p>NOTE: For Monarch Server 2021.1.1, the SDK and DataPrep versions are as follows:</p> <p>SDK version: 17.1.1.31052</p> <p>DataPrep version: 17.1.1.31052</p>
3	<p>Help</p> <p>Clicking this button opens a related page in the Administrator's Guide.</p>
4	<p>End Session</p> <p>Clicking this button logs you out of the application.</p>
5	<p><u>Navigation Tree</u></p> <p>This panel serves as the navigation point for Monarch Server.</p>

Panels and Functions	Description
	Click Show advanced options to see all available sections in the system.

NAVIGATION TREE

After [launching MSAdmin](#), at the left of the main window you see the **Navigation Tree**. It serves as navigation through the site. All items are expandable. It means that if you click a tree item or click an arrow by its name, the item will be expanded to the next tree level.

When you click a second-level option in the left pane, the right pane of the window displays the corresponding tab(s) with a list of items.

The Navigation Tree of MSAdmin contains links to the following sections of the system:

- ❑ **My Home:** This page is displayed by default, when you log on to MSAdmin. The right pane contains links to all sections of the system.
- ❑ Report Warehouse
 - Maintenance
 - ◆ **Log Clearance Tasks:** This link allows you to manage log clearance tasks. For more information, see [Log Clearance Tasks](#).
 - ◆ **Maintenance Processes:** This link allows you to manage processes. For more information, see [Maintenance Processes](#).
- ❑ Automator
 - **Server Status:** This link allows you to check the current status of the Monarch Server Automation Edition server. For more information, see [Server Status](#).
 - Standard Processes
 - ◆ **Processes:** This link allows you to handle Standard Processes. For more information, see [Processes](#).
 - ◆ **Schedules:** This link allows you to view schedules of Standard Processes. For more information, see [Schedules](#).
 - ◆ **Jobs:** This link allows you to manage Automator jobs. For more information, see [Jobs](#).
 - ◆ **Distributions:** This link allows you to specify distributions for items used in Standard Processes. For more information, see [Distributions](#).
 - ◆ **Alerts:** This link allows you to set up alerts on Standard Processes, jobs, and other items. For more information, see [Alerts](#).
 - ◆ **Scripts:** This link allows you to view scripts available for Standard Processes. For more information, see [Scripts](#).
 - ◆ **Project Editor:** This link allows you to create project files. For more information, see [Project Editor](#).
 - Visual Processes
 - ◆ **Processes:** This link allows you to handle Visual Processes. For more information, see [Processes](#).
 - ◆ **Templates:** This link allows you to access available templates that can be used for process creation. For more information, see [Templates](#).
 - ◆ **Logs:** This link allows you to view information about executed processes. For more information, see [Logs](#).
 - ◆ **Alerts:** This link allows you to set up alerts on Visual Processes. For more information, see [Alerts](#).

- ◆ **Scripts:** This link allows you to view scripts available for Visual Processes. For more information, see [Scripts](#).
- **Connection Definitions:** This link allows you to add predefined/saved connection definitions that can be reused in multiple projects. For more information, see [Connection Definitions](#).
- RMS
 - **Content Recognition Rules:** This link allows the automatic loading of models based on the content of the reports uploaded by the user. For more information, see [Content Recognition Rules](#).
- Portlets
 - **Portlets:** This link allows you to manage portlets. For more information, see [Portlets Management](#).
 - **Portlet Groups:** This link allows you to manage portlet groups. For more information, see [Portlet Groups](#).
- Server Library
 - **Folders:** This link allows you to create server library folders that include particular models, projects, and templates. For more information see [Folders](#).
 - **Projects:** This link allows you to add existing project files created with Monarch to the system and edit project files. For more information see [Projects](#).
 - **Models:** This link allows you to manage models. For more information, see [Models](#).
 - **Templates:** This link allows you to manage templates. For more information, see [Templates](#).
 - **Workspaces:** This link allows you to manage workspaces. For more information, see [Workspaces](#).
 - **Change Lists:** This link allows you to manage change lists. For more information, see [Change Lists](#).
 - **Bulk Edits:** This link allows you to apply text content bulk editing to Server Library items. For more information, see [Bulk Edits](#).
- Rights and Privileges
 - **Security Provider:** This link allows you to define a security provider. For more information, see [Setting Security Provider](#).
 - **Roles:** This link allows you to manage roles. For more information, see [Roles](#).
 - **User Groups:** This link allows you to manage user groups. For more information, see [User Groups](#).
 - **Users:** This link allows you to manage users. For more information, see [Users](#).
- System Settings
 - **System Preferences:** This link allows you to set system preferences. For more information, see [System Preferences](#).
 - **Locale Management:** This link allows you to manage [locales](#). For more information, see [Managing Locales](#).
 - **Bulk Schedules Edits:** This link allows you to apply bulk editing of schedules. For more information, see [Bulk Schedules Edits](#).
- News
 - **News Management:** When clicked displays three tabs: All News, Expired News and Global News. For more information, see [Working with News](#).
- System Reports
 - **Configuration:** This link allows you to generate configuration reports. For more information, see [Configuration Reports](#).
 - **Transaction:** This link allows you to generate transaction reports. For more information, see [Transaction Reports](#).
 - **Active Users:** This link allows you to monitor active users in the system. For more information, see [Active Users](#).

FUNCTIONAL BUTTONS

The following functional buttons are present throughout the application in the right pane:

-  Allows you to add a new item to the list displayed in the right pane.
-  Allows you to edit items (documents, document types, maintenance tasks and so on) from the list displayed in the right pane. Select the item to be edited, by clicking anywhere on a row, and click .

NOTE Instead of clicking this button you may click the name of an item in a list.

-  Allows you to remove items from the list displayed in the right pane.

To remove an item

1. Click  at the end of the row with the item you want to delete.
2. Click Delete to remove the corresponding item.

NOTE If an item does not contain any data, it will be removed from the list. If there are dependent items, you will not be allowed to delete it (for example, you cannot delete an index field if it is used for defining report types).

PAGE NAVIGATION

In MSAdmin, whenever an item list exceeds the [Count of Items Per Page](#) setting, items are listed on subsequent pages. You can access these additional pages via page navigation icons.

For example, let's say the specified **Count of Items Per Page** setting is 10. If there are 20 Monarch Server users and you attempt to view them on the Users page, only the first 10 users will be visible. At the bottom of the users page, the page navigation icons would appear like this:



To see the remaining 10 users, you would have to click one of the page navigation icons.

NOTE The two left icons are dimmed, because there are no preceding pages.

You can also change the number of items to be displayed on one page, by selecting an option (**10, 20, 50, 100, or 500**) from the **Items per Page** drop-down list.

To navigate between pages, type a page number in the **Page** box, at the bottom of the table, or use the page navigation buttons:

- To navigate to the next page, click .

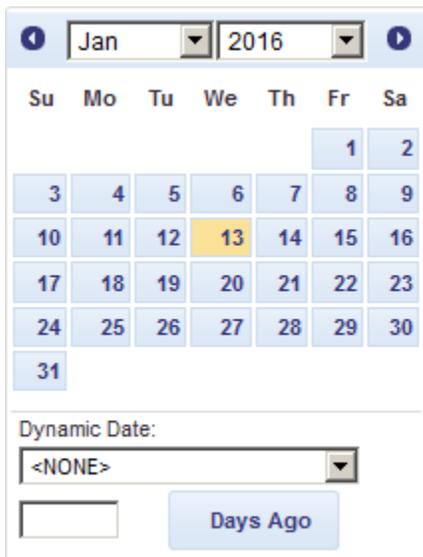
- ❑ To navigate to the last page, click .
- ❑ To navigate to the previous page, click .
- ❑ To navigate to the first page, click .

NOTE If there are no additional pages to view all page navigation icons are disabled. The current page is the only page available.

USING THE CALENDAR

When completing input information requirements of some pages you are required to define the date. To access the **Calendar** window, click the  icon next to a date field, or click the date field itself.

The **Calendar** window appears on your screen:



The screenshot shows a calendar interface for January 2016. At the top, there are navigation arrows and dropdown menus for the month (Jan) and year (2016). The calendar grid shows days from Sunday to Saturday. The date 13 is highlighted in yellow. Below the calendar, there is a 'Dynamic Date' dropdown menu currently set to '<NONE>', and a 'Days Ago' button next to an empty input field.

The left arrow navigates to the previous month, and the right arrow navigates to the next month.

To set a date

1. Select a month on the **Calendar** using arrows and then select the date.
2. Select one of the following predefined relative date values from the **Dynamic Date** list: Yesterday, End of Last Week, Start of This Quarter and so on.
3. Set a date as a number of days ago. Enter the corresponding value in the box and click **Days Ago**.

You can also use the following buttons, placed next to a date field:

- To decrease the date by one day, click .
- To enter the current date, click .
- To increase the date by one day, click .

REPORT WAREHOUSE

Report Warehouse allows end users to retrieve information from reports, search for data in several reports simultaneously, and manipulate data for further analysis. This online system makes report data available more quickly, economically, and effectively than is possible with traditional hardcopy reports.

The topics in this section provide information about:

- [Maintenance Management](#)

MAINTENANCE MANAGEMENT

The topics in this section provide information about:

- [Log Clearance Tasks](#)
- [Maintenance Processes](#)

Log Clearance Tasks

A log clearance task deletes records of a specified type from the Logs, Pump Job Logs, Schedule Jobs, and Visual Process Logs tables of the Monarch Server database. Use these tasks to delete old (or unnecessary) events from the database. They help to use database space economically.

To display the **Log Clearance Tasks** page, click **Report Warehouse** on the Navigation Tree, then click **Maintenance** and select **Log Clearance Tasks**. Use this page to [add](#), [edit](#), or [delete](#) log clearance tasks.

The following Event Source types of Logs table of log clearance tasks are available:

- Audit:** Select this option to delete audit events.
- Custodian:** Select this option to delete custodian events.
- Entity Audit:** Select this option to delete entity audit events.
- Filing:** Select this option to delete filing events.
- Maintenance:** Select this option to delete maintenance events.
- Pumping:** Select this option to delete pumping events.
- Pumping Job Log:** Select this option to delete pumping job log events.
- Security:** Select this option to delete security events.
- System:** Select this option to delete system events.

The following log statuses of the Pump Job Logs table of log clearance tasks are available:

- Cancelled:** Select this option to delete cancelled job log.
- Completed:** Select this option to delete completed job log.
- Failed:** Select this option to delete failed job log.

The following job/log statuses of Schedule Jobs and Visual Process Logs table of log clearance tasks are available:

- Completed:** Select this option to delete completed job log.
- Failed:** Select this option to delete failed job log.

NOTE Instead of deleting pump job log data, the log clearance task for pump jobs will mark pump jobs as deleted. [Custodian Schedule](#) should be used to completely delete pump jobs from the database.

Adding a Log Clearance Task

To add a log clearance task

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Log Clearance Tasks**.
2. Click  on the **Log Clearance Tasks** page.
3. **Maintenance Task Name:** Enter the name of a new log clearance task.
4. **Description:** Optionally, enter any text defining the clearance task.
5. **Date:** Specify the dates of events to be deleted by the current task in the **From ... To** boxes. Click  to [use the Calendar](#).
6. **Please, select Tables for clearing:** Specify the table to delete records from.
7. Select the event types or the job log statuses to delete from the specified table.
8. **Select All:** Click this button to select all options.
9. **Select None:** Click this button to clear the selection.
10. To save a new log clearance task, click **Save**.
If the operation is completed successfully, the system saves the a new task to the database and displays the updated **Log Clearance Tasks** page.
11. To clear the recent changes, click **Clear**.
12. To cancel the operation and return to the **Log Clearance Tasks** page without saving data, click **Cancel**. Keep in mind that no changes will be saved.

Editing a Log Clearance Task

To edit a log clearance task

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Log Clearance Tasks**. The Log Clearance Tasks page appears.
2. Click the name of the log clearance task you want to edit or click anywhere on its row, and then click . The Log Clearance Task page appears. For details about how to change its items refer to [Adding a Log Clearance Task](#).
3. Make the necessary changes and click **Save**. The system saves the modified log clearance task to the database and displays the updated **Log Clearance Tasks** page, if the operation is completed successfully.

Deleting a Log Clearance Task

To delete a log clearance task

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Log Clearance Tasks**.
2. Click  at the end of the row with the log clearance task you want to delete. A dialog box will appear.

3. Click **Delete** to delete the log clearance task from the list.

Running a Log Clearance Task

When you finish [configuring](#) a [log clearance task](#), you can run it to perform the specified action, in the same way as you assign a maintenance task.

To run a log clearance task

1. [Configure](#) a [maintenance process](#).

NOTE If the desired maintenance process already exists, skip down to step 2.

2. Assign the log clearance task to the maintenance process, in the same way as you [assign a maintenance task](#).
3. [Run the maintenance process](#). When the process is executed, the tasks assigned to it will be run as well.

Maintenance Processes

A maintenance process defines the schedule of executing maintenance tasks and log clearance tasks. All maintenance processes used in the system are listed on the Maintenance Processes page.

To display the Maintenance Processes page, click **Report Warehouse** on the Navigation Tree, then click **Maintenance**, and select **Maintenance Processes**.

The list shows:

- Process name
- Run Now (click  to run the process immediately)
- Suspend status
- Last started date and time
- Last finished date and time
- Next execution date and time of the current process

Use the **Maintenance Processes** page to [add](#), [edit](#), or [delete](#) a maintenance process.

You can also:

- Suspend all processes, by clicking the Suspend button below the list.
- Resume all processes, by clicking the Resume button below the list.
- Run a process immediately, by clicking the  icon next to its name.
- Update the status of maintenance processes by clicking the  icon, below the list.

With maintenance processes, you can schedule the execution of [maintenance tasks](#) to copy, move, delete, purge or re-index data. When a maintenance process runs, it executes all maintenance tasks that are assigned to it. When a maintenance process runs, it executes all log clearance tasks that are assigned to it. With maintenance processes, you can schedule the execution of [maintenance tasks](#) to copy, move, delete, purge or re-index data. When a maintenance process runs, it executes all maintenance tasks and log clearance tasks that are assigned to it. Maintenance processes can be [run](#) on a scheduled or manual basis.

Adding a Maintenance Process

[Maintenance processes](#) are used to schedule and run [maintenance tasks](#). [Maintenance processes](#) are used to schedule and run [log clearance tasks](#). When a maintenance process runs, it executes all maintenance tasks that are assigned to it.

To add a maintenance process

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Processes**.
2. Click  on the **Maintenance Processes** page. The Maintenance Process page appears.
3. **Name:** Enter the name of a new process.
4. **Number of concurrent tasks:** Enter the number of concurrent tasks. Enter **1**, if you want several tasks to be executed sequentially. Enter the actual number of tasks, if you want them to be executed simultaneously.
5. **Suspended:** Select this option to suspend the current process.
6. **Available Tasks:** Displays all available tasks, from which you can select tasks to assign to the current process. To do this, drag a task from the left pane to the right pane. The tasks are carried out in the order they appear on the list. You can change the order by dragging items up or down.
7. **Filter by:** The filter for the **Available Tasks** list. Type (part of) a task name, and click **Apply** to filter the list.
8. **Assigned Tasks:** The tasks in this pane will be assigned to the current process. To remove a task from this pane, drag it back to the left pane.
9. **Execution Frequency:** These options allow you to set the periodicity of launching the current process. For more information, see [Execution Frequency](#).
10. To save a new process, click **Save**.
The system saves the new process to the database and displays the updated Maintenance Processes page, if the operation is completed successfully.
11. To allow an administrator to launch the process immediately (subject to internal processing queues) and save the schedule in the database with the **Run Once** scheduling type and the current date as a date to run, click **Run Now**.
12. To clear the recent changes, click **Clear**.
13. To cancel the operation and return to the **Maintenance Processes** page without saving data, click **Cancel**.
Keep in mind that no changes will be saved.

Execution Frequency

Execution frequency (or run time) specifies both how often a process will run (for example, once, hourly, daily, etc.) and at what time it will run. You specify execution frequency when configuring a [filing](#) or [maintenance process](#).

The options in the Execution Frequency pane allow you to set periodicity of launching the current process. An administrator can set the following options:

- One Time:** Select this option if you want to launch the current process only once.
 - **At:** Set the time when the current process will be launched following the HH:MM format.
 - **on (mm/dd/yyyy):** Set the date when the current process will be launched. Upon clicking the field, the [calendar](#) appears.
- Hourly:** Select this option if you want to launch the current process at the specified hour or hours.
 - **At... minute(s) of Every hour:** Set the time when the current process will be launched following the MM format.
- Daily:** Select this option if you want to launch the current process in a specified number of days.
 - **Every ... day(s):** Set the number of days after which the current process will recur.
 - **At:** Set the time when the current process will be launched following the HH:MM format.

- ❑ **Weekly:** Select this option if you want to launch the current process weekly.
 - Recur every: Set the number of weeks after which the current process will recur.
 - Select corresponding days of the week.
 - At: Set the time when the current process will be launched following the HH:MM format.
- ❑ **Monthly:** Select this option if you want to launch the current process monthly.
 - Select the day when to perform the process.
 - ◆ If you select **Day**, set the number of days in the box.
 - ◆ If you select **Business Day**, set the business day(s) on which the current process will be launched. Business days are weekdays (i.e., Monday-Friday), unless an exception date is defined. The definition of [Exception Dates](#) can be used for holidays. When an exception day is found, the count skips to the next working day.

The process will be executed immediately after exception date/time is over. Otherwise, the process will only run on the set schedule, with the exception of the Business Day schedule, wherein an “All day” Exception Date will be excluded in the day count.

Examples:

To run the process on the 6th business day of the month, enter “6” in the field.

To run the process on the 1st business day of the month, then again on the 16th to the 20th business day of the month, enter “1, 16-20” in the field.

NOTE

It is possible to schedule the process on a range of working days at a given time, e.g., “5”; “1,2,22,23”; “1-3, 21-23”; “-1”; “-3--1”; and “-2-3, 5, 10.”

A negative number denotes the number of days prior to the month selected, e.g., “business day -1” of October 2019 would be September 30, 2019 (provided it is a working day). Exception Dates must skip count to the next previous working day.

- ◆ If you select **The...**, select First, Second, Third, Fourth or Last from the first drop-down list and the day of the week from the second drop-down list.
- Select the month on which to perform the action.
 - ◆ If you select the **of every n Month** option, set the number of month in the box.
 - ◆ If you select the **of selected months** option, select the appropriate check box referring to the desired month.
- In the **At** check box set the time when the current process will be launched following the HH:MM format.

Assigning Maintenance Tasks to Maintenance Processes

To execute a [maintenance task](#), you need to assign it to a [maintenance process](#). The process runs and executes the task at a certain time, according to the scheduled execution frequency.

NOTE

You can [configure a new maintenance process](#) and add the maintenance task to it, or you can add the task to an existing process.

To assign a maintenance task to a maintenance process

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance**, then select **Maintenance Processes**.
2. Click the name of the maintenance process you want to add the maintenance task to or clicking anywhere on its row, and then click .
3. From the **Available Tasks** pane, drag the necessary task to the **Assigned Tasks** pane.
4. Set the process [execution frequency](#).
5. Click **Save**.

The process will execute the maintenance task according to the scheduled execution frequency.

NOTE If you prefer, you can execute the process right away by clicking the **Run Now** button.

Running a Maintenance Process

When a [maintenance process](#) runs, it executes all [maintenance tasks](#) assigned to it. When a [maintenance process](#) runs, it executes all [log clearance tasks](#) assigned to it. A maintenance process can be run in two ways:

Manually: To run a maintenance process manually

1. Click the  icon next to its name on the [Maintenance Processes page](#) or click the **Run Now** button at the bottom of the [Maintenance Process page](#). You can do this immediately after configuring a new process, or you can manually run a previously configured process:
 - a. click **Report Warehouse**, then click **Maintenance Management** on the Navigation Tree and select **Maintenance Processes**.
 - b. Click the name of the process you want to run.
 - c. Click the **Run Now** button at the bottom of the Maintenance Process page.

NOTE When you click the **Run Now** button, there may be a brief delay before the process actually runs. This is likely due to a time discrepancy between the server's clock and the clock on a local machine. Processes are run according to the time of the server's clock.

Scheduled: To execute a maintenance process via a scheduled runtime

1. [Specify the execution frequency](#) for it on the **Maintenance Process** page, then click **Save**. The process will be executed according to the scheduled execution frequency.

Editing a Maintenance Process

[Maintenance Processes](#) can be edited via the Maintenance Process page.

To edit a maintenance process

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Processes**.

2. Click the name of the maintenance process you want to edit or click anywhere on the row, and then click . The Maintenance Process page appears. For details about how to change its items refer to [Adding a Maintenance Process](#).
3. Make the necessary changes and click **Save**. The system saves the modified process to the database and displays the updated Maintenance Processes page.

Deleting a Maintenance Process

If you no longer need a [maintenance process](#), you can delete it.

To delete a maintenance process

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Processes**.
2. Click  at the end of the row with the maintenance process you want to delete.
3. Click **Delete** to delete the maintenance process from the list.

AUTOMATOR

Automator is a data conversion solution that uses existing ASCII or ANSI reports, PDF files, XPS files, ISAM files (e.g. XLS, MDB, ACCDB, DBF, Open XML Spreadsheets (XLSX/XLSM), html files, and OLEDB/ODBC sources as a source for data. You can input report files (e.g., TXT, DAT, PRN, LOG, .etc.) into Automator and then export their data to a variety of formats (e.g., PDF, XLS, XLSX/XLSM, MDB, ACCDB, HTML, DBF, etc.) or to an OLE DB or ODBC destination. Automator also allows you to export files to a Microsoft Office SharePoint Server, FTP/SFTP site, email destination, or distribution list. In short, Automator lets you extract, transform, move and load data faster and more easily than previously possible.

Features and Functions

Automator has the following capabilities:

- Automated conversion of data from TXT, PDF, ISAM, XPS, and OLE DB/ODBC sources to an ISAM, or OLE DB/ODBC (e.g. SQL Server, Oracle) target by scheduled or event-driven execution via monitored folders.
- Email distribution of PDF, ISAM, and Open XML Spreadsheets
- File system and SharePoint distribution of input and export files
- RSS publication
- Visual Process Designer tool
- Creating logs and alerts resulting from operations
- Allowing user creation of pre-processes and post-processes using VB.NET Scripting
- Scheduled administration of job logs and database entries

SERVER STATUS

The **Server Status** page displays information detailing the current status of Automator.

To view the **Server Status** page, click **Automator** and select **Server Status**.

The **Server Status** page includes the following information:

- Refresh Status
 - **Interval** (in seconds): Specify the refresh status interval in seconds.
 - **Refresh Now**: Click the button to refresh status now.
 - **Automatic**: Select this check box to refresh status at the specified interval automatically.
- Server Status
 - **Info from database**
 - ◆ **Standard Processes**
 - **Custodian Status**: Shows whether the custodian schedule is running.
 - **Jobs/Processes Currently Executing**: Shows the number of jobs currently being executed. Hover over the  icon to view the names of the jobs.

- **Process Queue Entries:** Shows the number of Standard Processes in the queue that have not been picked up by the dispatcher. Click the Delete button  to clear the current queue of Standard Processes. Hover over the  icon to view the names of the processes in queue.
- **Active Jobs/Processes:** Shows the number of active jobs. Hover over the  icon to view the names of active jobs.
- **Process Count:** Shows the total number of processes in the system.
- **Jobs/Logs Count:** Shows the total number of jobs in the system.
- ◆ **Visual Processes**
 - **Jobs/Processes Currently Executing:** Shows the number of Visual Processes currently being executed. Hover over the  icon to view the names of the processes being executed.
 - **Process Queue Entries:** Shows the number of Visual Processes in the queue that have not been picked up by the dispatcher. Click the Delete button  to clear the current queue of Visual Processes. Hover over the  icon to view the names of the processes in queue.
 - **Active Jobs/Processes:** Shows the number of active Visual Processes. Hover over the  icon to view the names of the active processes.
 - **Process Count:** Shows the total number of processes in the system.
 - **Jobs/Logs Count:** Shows the total number of logs in the system.
- **Info from runtime**
 - ◆ **Standard Processes**
 - **Jobs/Processes Currently Executing:** Shows the number of jobs currently being executed.
 - **Active Jobs/Processes:** Shows the number of active jobs.
 - ◆ **Visual Processes**
 - **Jobs/Processes Current Executing:** Shows the number of Visual Processes currently being executed.
 - **Active Jobs/Processes:** Shows the number of active Visual Processes.

NOTE The database and runtime values should be ideally equal. However, in some cases, the database values may not be updated immediately once a job or process is executed. In this scenario, the database and runtime values will not be the same.

- **The number of processes in the system:** Shows the number of processes that have been reached but have NOT exceeded the number allowed by the license.

NOTE If the number of processes in the system exceeds the number allowed by your license, the following message appears instead:

“The maximum number of allowed processes per your license (XX) is exceeded.”

The number of processes is the total count of Visual and Standard Processes.

- **CPU Cores Usage:** Shows the number of CPU cores available for the system.

NOTE When the number of CPU cores allowed by your license is exceeded, the following message appears:

“The maximum number of allowed CPU cores per your license (XX) is exceeded.”

Here, “XX” is the number of CPU cores allowed by your license.

- ❑ **Dispatcher Status:** Displays a summary of the server status, including the server name, number of running jobs/Visual Processes, and number of jobs/Visual Processes in queue.
- ❑ Operations
 - **Job Processing Status:** Shows the current job processing status.
 - **Start:** Click the button to start the Automator Dispatcher Service Host. Starting dispatcher enables running of new jobs.
 - **Stop:** Click the button to stop the Automator Dispatcher Service Host. Stopping dispatcher disables running of new jobs.

NOTE When the **Job Processing Status** is disabled, a notification in the Admin interface and the logs would indicate the disabled status of the Automator Dispatcher Service Host.

STANDARD PROCESSES

This section specifies how to use various project items supported for the previous versions of Automator and Datawatch Data Pump products. These earlier versions of products use the standard approach for handling with processes, schedules, jobs, alerts, etc.

The **Standard Processes** includes the following items:

- ❑ [Processes](#)
- ❑ [Schedules](#)
- ❑ [Jobs](#)
- ❑ [Distributions](#)
- ❑ [Alerts](#)
- ❑ [Scripts](#)
- ❑ [Project Editor](#)

Processes

A process contains a set of one or more XML project files (for example, XPRJ or DPRJ files) that are to be executed to perform a desired Automator operation. Processes utilize Monarch models and project files to extract data from text or database sources, and export it to various file types, such as, XLS, MDB, TXT, HTML, and XML.

Processes are configured on the **Processes** page of the Automator Standard Processes section. When adding a process, you name it, assign a project file or files to it, and set a schedule for when it is to be run.

All existing system processes are displayed on the **Processes** page. To view the **Processes** page, on the **Navigation Tree**, click **Automator > Standard Processes** and select **Processes**.

On this page you can view the following information:

- Process name
- Jobs column linked to the Jobs page
- Process description
- Process priority
- Usage column showing the current status of the process:
 - Monitored enabled
 - Monitored disabled
 - Scheduled enabled
 - Scheduled disabled
 - Ad hoc (i.e., no monitoring or schedule defined)
 - Report Warehouse Input enabled
 - Report Warehouse Input disabled

Use this page to:

- [Add a process](#)
- [Edit a process](#)
- [Schedule a process](#)
- [Run a process](#)
- [Delete a process](#)
- [Import a process](#)
- [Export a process](#)

To re-run a process, click the **Run Now** icon  .

NOTE Using the corresponding fields in the column headings, you can filter the processes by

- Name
- Description
- Priority
- Usage

The following filter settings are available:

- ~ - contains
- == - equal
- < - less
- > - greater
- X - Reset search value

Adding a Process

With Automator, you can configure processes to extract data from text or database sources and to export it to various file types (for example, XLS, MDB, TXT, HTML, XML).

To add a Process

1. On the Navigation Tree, click **Automator** and select **Standard Processes**.
2. On the **Processes** page, click the **Add** icon . The page with general process settings will be displayed.

NOTE If you exceed the number of processes allowed by your license, the following message appears:

"The license for Monarch Server is used improperly. The maximum number of processes per your license (XX) is exceeded as you have NN processes currently defined and no more processes can be defined. You may delete unused processes or contact your Altair sales representative for extended options."

3. In the Name field, enter the name you want to assign to the process.
4. In the Description field, enter an appropriate description of the process.
5. Specify the number of projects per job:
 - **One project per job:** Select this option, to specify one project per job.
 - **Multiple projects per job:** Select this option, to specify several projects per job.

NOTE If you select the **One project per job** option, each project assigned to the process will spawn its own job. If you select the **Multiple projects per job** option, all projects will be executed as one job. See [Process Handling Scenarios](#) for more information.

6. In the **Location** field, select the location type from the drop-down list.

NOTE If the required server is not on the list, refer to [Locations Management](#) topic to add a new location. In case of a single server configuration, refer to the Configurator Guide in the [Monarch Server Installation Guide](#) to change location settings.

If a process cannot be executed on the specified location (e.g., the server is unavailable, the server has high utilization, or the Automator Dispatcher module is not installed), then it will be executed on any available location with the least utilization.

7. In the **Project File Paths** box, specify at least one project file in the .xprj or .dprj format. For this click the **Add** button and the **Add Project File Path** window appears. In the displayed window do one of the following:

- In the Project File Path field, type the path to the project file. Click Save to add the project file.
- From the the Source Type list, select the appropriate data source type and depending on the data source type specify other parameters:
 - ◆ If Named Paths selected, specify the [named paths](#) and choose the project file path from the list of displayed in the Named Path Content box. Click Save to add the project file.
 - ◆ If Server Library selected, specify the Server Library types and choose the Server Library from the list of displayed in the Available Server Library box. Click Save to add the project file.

NOTE If you assign more than one project to the process, specify the order in which you want them to be processed. Projects in the list are processed from top to bottom.

8. Click the **Save** button to save the project or **Run Now** to run the project immediately. The system saves the new process to the database and displays the updated Process page, if the operation is completed successfully.

Editing a Process

Once you have configured and saved a process, you can edit it at any time.

To edit a process

1. On the Navigation Tree, click **Automator** and select **Standard Processes**.
2. Click the name of the process you want to change or clicking anywhere on its row, and then click .
3. When you open a saved process for editing, the following additional tabs become available on the Process Properties page:
 - **General** tab: Allows you to [specify general settings](#).
 - **Advanced** tab: Use this tab to [specify advanced settings](#).
 - **Job Disposition** tab: Use this tab to [specify job log disposition settings](#).
 - **Monitoring** tab: Allows you to [specify input file monitoring settings](#).
 - **Script** tab: Allows you to add, view, or edit a [process-specific script](#).
 - **Schedule** tab: Allows you to [specify process schedule settings](#).
 - **Rights** tab: Allows you to assign process access rights to other users, roles or groups.
 - **Alerts** tab: Allows you to define process alerts that will only act for the current process. Process alerts act in addition to global alerts. Adding and use of process alerts is identical to global alerts adding. Refer to [Alerts](#) section for details.

Specifying General Settings

For details about how to change general settings please refer to [Adding a Process](#).

After specifying general settings for the process, you can click the **Advanced** tab to specify [advanced settings](#) or you may click the **Save** button to save the general settings and close the page.

Specifying Advanced Settings

After you have created a process you can specify its priority, whether to allow parallel jobs, and whether to enable process retries.

To specify advanced settings for a process

1. On the Navigation Tree, click **Automator** and select **Standard Processes**.
2. Click the name of the process you want to change or click anywhere on its row, and then click .
3. On the displayed page, select the **Advanced** tab.
4. In the **Job Priority** field, specify the priority for the job.
When multiple processes are launched at the same time, the priority settings primarily determine the order they run. Other factors may affect the priority of a process. Select the priority from the following options:
 - Lowest
 - Below Normal
 - Normal
 - Above Normal
 - Highest
5. Select the **Allow Parallel Jobs** check box, if you want to allow parallel jobs.
When parallel jobs are enabled, Automator creates a job for each of the input files assigned to a process. This setting only applies to processes that contain multiple input files when the One Project Per Job option is selected on the General tab. Enabling parallel jobs should be used in conjunction with an output [naming macro](#) that ensures the creation of unique file names throughout the duration of the process, for example, the Job Number macro.

NOTE The order of the projects in the process will not be observed, as all jobs will execute at the same time.

6. The **Process Priority** option is used for monitoring. If a number of processes have monitoring enabled and they are launched simultaneously, then processes with the highest priority run first. Any number from 1 to 255 is a valid process priority setting. Processes run in ascending numerical order according to their priority settings (for example, a process with the priority of 1 runs before a process with the priority of 2 or higher).
7. Set the behavior of the process when exporting to .accdb, .dbf, and .mdb in the **Export Behavior for accdb/dbf/mdb** option. The following options are available:
 - **Default** — Use this option for all new processes.

NOTE The default behavior can be defined in the config.xml file as follows:

```
<ServerModule
Name="Datawatch.RemoteServices.DataPumpDispatcherServiceHost"
MaxThreads="10">
  <AppSettings>
    <![CDATA[
<configuration>
  <appSettings>
    <add key="SdkEnableAceDaoExportByDefault"
value="true" />
<!-- use "true" for ACE DAO or "false" for ACE OleDb values-->
    </appSettings>
</configuration>
]]>
  </AppSettings>
```

```
</ServerModule>
```

- **ACE DAO** - Use this option for better performance for large datasets. All exports in the system with this behavior will be executed concurrently. ACE OleDb will be used as fallback when output tables exist option is "Update existing rows" or "Update existing rows and append new."
- **ACE OleDb** - Use this option for better reliability. This behavior is slower than ACE DAO, but it allows to handle exports in parallel.

NOTE The values for the **Export Behavior for accdb/dbf/mdb** option can be edited in [Bulk Edits](#) for several processes at the same time.

Take for example two jobs from different processes in a queue that exports to a different mdb.

- If the export behavior is set to ACE DAO, then the second job will wait for the first job. The second job just wastes a free slot in the ThreadPool.
- If the export behavior is set to ACE OleDb, then the two jobs will run in parallel.
- In summary, two jobs in parallel will be completed in ~10 min with ACE DAO and ~20 min with ACE OleDb.

In case of five jobs in parallel, then they will be completed in ~25 min with ACE DAO and ~20 min with ACE OleDb.

8. Select the **Bulk Behavior for OleDb** check box to enhance performance of table-based exports only (not summary).

NOTE This option is applicable for Oracle Database and Microsoft SQL Server OLEDB providers only.

9. Select the **Use Transaction** check box to revert the export via rollback if an error occurs during the export.

- NOTE**
- The **Use Transaction** check box is only available when the **Bulk Behavior** check box is selected.
 - Selecting this option will affect the performance of the table export and may require a large amount of system resources, especially for the target destination. Therefore, make sure to test this option first and then make the necessary adjustments.

The transaction is valid for the export operation only and not for the overall process.

10. Select the **Retry automatically if process prerequisites are not satisfied** check box, if you want to enable process retries.
When configuring a process, one of the settings you can specify is a process retries setting. This setting allows a process to be retried if it fails for some reason, such as due to a process prerequisite not being met. When specifying a process retries setting, you may specify the interval for the retry in hours and minutes, as well as the number of retries to attempt.
The retry functionality can be useful in performing a type of polling, as well as dealing with delays in the arrival of files necessary for a process. For example, if you wish to poll for the existence of a file, you could set up a schedule that operates every 5 minutes and then set a retry that has an interval of 1 minute which retries 4 times.
11. Specify the desired retry interval in hours or minutes in the **Retry every n hours n minutes** field.
12. Select one of the following options:
 - **Continue retrying for:** Select this option to specify the desired retry duration in minutes, hours and/or days.
 - **Continue retrying indefinitely:** Select this option to continue retrying the process indefinitely until it successfully runs.
 - **Retry at most n times:** Select this option to enter the maximum number of times you want Automator to retry the process.
13. Select the **Once found check n times for consistent file size** check box to specify that a file size check be carried out once the prerequisite is found. Also enter the number of times the check is to be carried out.
This allows the system to cope with file inputs from slow, unreliable or multi-stage write sources. Automator will test the input file, ensuring the file size has not changed for a specified period to ensure that the file is fully ready to be processed.
14. Specify a retry interval for checking the file size in the **Retry every n minutes**.
15. Select the **Retry automatically if job export failed** check box, if you want to enable failed export retries. This option allows to restart the process export provided it is blocked. When specifying an export retries setting, you can specify the interval for the retry in minutes and seconds, as well as the number of retries to attempt.
16. Specify the desired retry interval in minutes or seconds in the **Retry every n minutes n seconds** field.
17. Select one of the following options:
 - **Continue retrying indefinitely:** Select this option to continue retrying the failed export indefinitely until it successfully runs..
 - **Retry at most n times:** Select this option to enter the maximum number of times you want Automator to retry the failed export.
18. Click the **Save** button to save the current settings. The system saves the changes and displays the updated Process page, if the operation is completed successfully.

Specifying Job Log Disposition Settings for a Process

Job log disposition settings specify what to do with the job log after the job is finished, and under what circumstances the specified disposition will occur. Job log disposition settings can be specified on a process basis or you can specify default job log disposition settings, which will apply to all job logs.

NOTE After you specify job log disposition settings for a [process](#), be sure to specify a schedule for the [Custodian](#) and enable it, as the Custodian is responsible for implementing the disposition settings. If you do not specify a schedule for the Custodian, the disposition settings will not be implemented.

To specify job log disposition settings for a process

1. On the Navigation Tree, click **Automator** and select **Standard Processes**.

2. Click the name of the process you want to change or click anywhere on the row, and then click .
3. On the displayed page, select the **Job Disposition** tab.
4. Select one of the following Job Log Disposition actions:
 - **Leave:** Leaves the job log in its current location and with its current name.
 - **Remove:** Removes the job log.
 - **Move:** Moves the job log to the location you specify.

You can either enter the destination folder path in the text box or select the folder by clicking the **Browse** button. The **Named Paths** dialog box will appear.

- From the Named Paths list, select the appropriate [named path](#).
- Select the folder from the list of displayed in the Named Path Content box.
- Click **Save** to add the folder.

NOTE You can also use a [naming macro](#) to assign a new name to the job log by selecting one from the drop-down list. To do so, click the **Insert Macro** icon  and select the desired macro.

5. Select one of the following Apply Disposition By Job Status settings:
 - **Completed:** Select this check box, if you want the specified disposition to occur when a job is successfully completed. Use the adjacent fields to specify the number of days and/or hours after the job is completed that you want the disposition to occur.
 - **Failed:** Select this check box, if you want the specified disposition to occur when a job fails to run successfully. Use the adjacent fields to specify the number of days and/or hours after the job fails that you want the disposition to occur.
 - **Cancelled:** Select this check box, if you want the specified disposition to occur when a job is cancelled. Use the adjacent fields to specify the number of days and/or hours after the job is cancelled that you want the disposition to occur.
6. Click the **Save** button to save the current settings and close the dialog.

Specifying Input File Monitoring Settings for a Process

When you specify input file monitoring settings for a process, you specify input files that you want Automator to monitor a particular location for. When the file arrives in the specified location, Automator will trigger running of the associated process.

You can specify the input file monitoring settings on the **Monitoring** tab of the Process page.

To specify input file monitoring settings

1. On the Navigation Tree, click **Automator** and select **Standard Processes**.
2. Click the name of the process you want to change or click anywhere on its row, and then click .
3. On the displayed page, select the **Monitoring** tab.
4. Select the **Enable** check box to enable monitoring.

NOTE Changes in monitoring settings, including enabling and disabling monitoring or changing paths for monitored folders, take effect after approximately 1 minute.

5. Select the monitoring type.
 - Choose **Inputs** if the file to be monitored is an input file. Proceed to **Step 6**.
 - Choose **Manifests** if the file to be monitored is an XML file. Proceed to **Step 7**.
6. Click the **Load File Specifications from Project** button to specify the way specifications will be added from the following:
 - Append to the end of the list, or
 - Replace the list
7. Click the **Add** button to select the folder you want Automator to monitor for reports. The **Add File Specification** dialog box will appear.
8. Select one of two options in Manifest PostProcessing.
 - Delete (selected by default): Removes the XML file after processing
 - Move (and set the folder path): Moves the XML file after processing
9. In the **Add File Specification** dialog box, specify the following parameters:
 - **File Specification:** Enter the file path with the name of the file you want Automator to monitor for at the end, or select the file from the [Named Paths](#) list.

NOTE If you're unsure of the file's exact name or if you want Automator to monitor for any file of a particular file type, you can use a wildcard character, either an asterisk (*) or a question mark (?), so that the arrival of any file of the specified type will trigger the running of the process. An asterisk may replace multiple characters while a question mark may replace any single character.) For example, adding *.prn to the end of the folder path instructs Automator to trigger the running of the associated process when any report file with a .prn extension arrives in the specified location.

- **Location:** Select the location type from the drop-down list.

NOTE If the required server is not on the list, refer to [Locations Management](#) topic to add a new location.

- From the Named Paths list, select the appropriate [named path](#), and select the project file from the list displayed in the Named Path Content box.
 - Click **Save** to add the file.
10. Back in the **Monitoring** tab, specify the monitoring behavior:
 - **Disable monitoring if trigger file is not removed** – The input mask (e.g., myfile.txt or *xyz.pdf) from the monitored folder can sometimes cause a process job to fail with an error and remain in its original folder. By selecting **Disable monitoring if trigger file is not removed**, Automator stops monitoring the corresponding folder(s) for the process and its input mask(s) only. When this option is not selected, the monitoring is left enabled; when new files appear in the monitoring folder, the process running is triggered again. You would, for example, enable this option if a problem requires intervention to determine why the inputs caused the process to fail and prevent the process from running until the issue is addressed.

- **All files must exist** – Monitoring based on multiple files starts as soon as one of the files is found. By selecting the *All files must exist* option, the monitoring process will not start unless all the associated files are found.

11. Choose to download a manifest file that is based on the settings of the standard process. In **Download Manifest**, choose between the following:

- **XML** – creates a manifest XML file
- **JSON** – creates a manifest JSON file

NOTE Before starting a process with a manifest containing some passwords, make sure to specify all required passwords and uncomment the “TODO:Specify Password” row in the manifest. If the process has no passwords, the manifest file generated automatically could be retained as it is.

Click the **Download** button, and rename and save the manifest file as usual.

12. Click the **Save** button to save the input file monitoring settings and proceed to the Processes page.

NOTE When you specify manifests monitoring settings for a process, you specify the manifest XML files that you want Automator to monitor a particular location for. When the file arrives in the specified location, Automator will trigger running of the associated process.

Specifying Scripts for a Process

On the **Script** tab you can create and edit the [global environment](#) and [process scripts](#).

The following scripts are displayed on the page:

- [PreProcess tab](#): Specifies process-specific, pre-process declarations.
- [PostProcess tab](#): Specifies process-specific, post-process declarations.
- [Global Declarations tab](#): Specifies Dim statements or other statements that must be global to the entire script module.
- [References and Imports tab](#): Specifies additional .NET assembly references and namespaces required by the PostExport, PreExport, and Global Declarations tabs.
- [Passwords tab](#): Specifies passwords for use in password macros.

NOTE Automator users can only edit these scripts if they are granted editing privileges by the Admin.

Scheduling a Process

After adding a process, you can create a process schedule to specify when, and at what interval the process will run. All schedules used in the system are listed on the [Schedules](#) page. You can [add](#), [edit](#) or [delete](#) the schedule on the **Schedule** tab of the process page.

To open the **Schedule** tab

1. On the Navigation Tree, click **Automator** and select **Standard Processes**.
2. Click the name of the process you want to change or clicking anywhere on its row, and then click .
3. On the displayed page, select the **Schedule** tab.

To suspend the selected process

1. Click the **Suspend Selected** button.

To resume the selected process

1. Click the **Resume Selected** button.

To suspend all processes

1. Click the **Suspend All** button.

To resume all processes

1. Click the **Resume All** button.

NOTE If the process fails to run, see [When a Job Fails to Run](#) for possible explanations.

Adding a Schedule

A schedule is used to specify when, and at what interval the process will run.

To add a new schedule for a process

1. On the **Schedule** tab, click .
2. Select the **Suspended** check box if you want to suspend the task.
3. Select the task [execution frequency](#) from the following options: **One Time, Hourly, Daily, Weekly, Monthly**.
4. Set the time you want the process to run.
5. Set the start date of the process.
6. Select the appropriate timezone from the drop-down list.
7. Click the **Save** button, to save the schedule or **Run Now** to start the task immediately.

Editing a Schedule

For details about how to change the schedule settings please refer to [Adding a Schedule](#).

Deleting a Schedule

You can delete any schedule on the Schedule page.

To delete a schedule

1. On the Navigation Tree, click **Automator** and select **Standard Processes**.
2. Click the name of the process you want to change or click anywhere on its row, and then click .
3. On the displayed page, select the **Schedule** tab.
4. Click  at the end of the row with the schedule you want to delete. A dialog box appears.

5. Click **Delete** to remove the schedule from the list.

Assigning Process Access Rights

The creator of a process automatically receives the rights to view and edit the project.

You can assign process access rights to other users. Authorized users can view and edit processes, assigned to them, in MSAdmin.

To assign process access rights

1. On the Navigation Tree, click **Automator** and select **Standard Processes**.
2. Click the name of a process you want to assign access rights for.
3. On the displayed page, select the **Rights** tab.
4. To make a process available to all users, select **Global** check box and click the **Save** button. Otherwise proceed to the next steps.
5. Click **User Groups** to expand a hidden section. Select user groups that you want to have an access to the process.
 - a. To add a user group drag it from the left pane to the right pane or click  icon next to its name.
 - b. To remove a user group drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

6. Click **Roles** to expand a hidden section. Select roles that you want to have an access to the process.
 - a. To add a role drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a role drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

7. In the **Users** section, select users that you want to have an access to the process.
 - a. To add a user drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a user drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

8. To apply the changes, click **Save**.

Deleting a Process

You can delete any process on the **Processes** page.

NOTE When you delete a process, all of its associated jobs on the Jobs page are also deleted.

To delete a process

1. On the Navigation Tree, click **Automator** and select **Standard Processes**.
2. Click  at the end of the row with the process you want to delete. A dialog box appears.
3. Click **Delete** to remove the process from the list.
The process is removed from the [Processes](#) page, and its associated jobs are removed from the [Jobs](#) page.

NOTE When deleting a standard process that uses a model or a project in the Server Library, a dialog box appears and gives the option to **Delete dependencies**. Clicking the **Delete dependencies** button removes the associated model or project from the Server Library. Clicking **Cancel** deletes the process only and not the associated model or project. The **Delete dependencies** option is not available when the model or project is used by more than one process.

Enabling/Disabling Monitoring of Standard Processes

You can enable or disable the monitoring of multiple Standard Processes simultaneously.

To enable/disable process monitoring

1. On the Navigation Tree, click **Automator**, and select **Standard Processes**.
2. Select the Standard Processes you want to enable/disable the monitoring for by clicking their corresponding checkboxes.
3. Click the drop-down arrow , and choose **Disable Monitoring** or **Enable Monitoring**.
4. In the **Monitoring** dialog box, click **Ok** to proceed.

NOTE When enabling/disabling monitoring or changing the path of a monitored folder, the changes take effect after approximately 1 minute.

Enabling/Disabling Scheduled Standard Processes

You can enable or disable one or more scheduled Standard Processes.

To enable/disable scheduled Standard Processes

1. On the Navigation Tree, click **Automator**, and select **Standard Processes**.
2. Select the Standard Processes whose schedule you want to enable/disable by clicking their corresponding checkboxes.
3. Click the Additional schedule actions icon , and choose **Disable Schedules** or **Enable Schedules**.
4. In the **Monitoring** dialog box, click **Ok** to proceed.

NOTE When enabling schedules, the corresponding **Suspend** checkbox in **Schedules** is unchecked (i.e., the schedule is active). When disabling schedules, the corresponding **Suspend** checkbox is checked (i.e., the schedule is suspended)

Running a Process

Automator provides several methods by which you can run a process.

Process can be run:

- ❑ **Manually:** You can run a process manually by clicking the Run Now icon  in the corresponding row. Automator will then generate and execute a job (or jobs) for the process.

NOTE If the Config service is not available and you are using Altair licensing, Automator processes will be switched to "Limp Mode." In this mode, you can run only one process (i.e., one Standard Process and one Visual Process) every hour. Moreover, you cannot run a Visual Process that involves other Standard or Visual Processes. Limp Mode is deactivated once the Config service becomes available.

- ❑ **Via a [process schedule](#):** When you configure a process schedule, you specify a day and time for the process to run.

NOTE You can specify multiple schedules for a process if desired.

- ❑ **Via an [input file specification](#):** An input file specification instructs Automator to monitor a particular location for a certain input file (or for any file of a specified file type), and to trigger the running of the associated process when the input file arrives there.

NOTE If a corresponding job for a process fails to run, refer to [When a Job Fails to Run](#). You should also view the [job's log](#) file, as they may indicate why the job failed.

Importing a Process

You can import a process from the **Processes** page.

You can only import a process by selecting the output (in XML format) of a previous export process operation.

To import a process

1. On the Navigation Tree, click **Automator**, and select **Standard Processes**.
2. On the displayed page, select the **Import Processes** tab.
3. In the **Location** field select the location type from the drop-down list.

NOTE If the required server is not on the list, refer to [Locations Management](#) topic to add a new location.

4. Click the **Add Files...** button and choose the file(s) to import in the **Open** dialog box. You can select several files at once. Alternatively, you can drag your files to the **Drop files here** area.
5. Enable the **Include process dependencies** check box to import projects and project models connected with the processes (if any) to Server Library. If a project/model is unavailable in the file system, you can opt to upload the new one instead.
6. Click **Save** to save the process. A message box will inform you that the import operation is complete and the [Processes page](#) will be displayed.

NOTE You cannot specify location for the processes exported with a defined location.

If you exceed the number of processes allowed by your license, the following message appears.

"The maximum number of processes per your license (XX) will be exceeded with this import and it cannot be completed. You can select at most (NN) processes for import, or you may delete unused processes or contact your Altair sales representative for extended options."

Named Path Mapping

During import of an exported Standard Process that includes named paths, the **Named Path Mapping** window displays. In this window, the named paths in the Standard Process being imported are mapped to the existing named paths in the destination environment. The following options are available:

-  Auto Map Named Path button – Click to automatically map all imported named paths by ID or by Name (depending on the radio button selected). By default, the imported named paths are mapped by Name.
-  Edit Mapping button – Click to manually perform mapping or modify existing mapping.
-  Delete Mapping button – Click to delete currently selected mapping.
-  Delete All Mappings button – Click to delete all listed mappings.

Exporting a Process

You can export a process on the **Processes** page.

The output of the export operation will be an XML file (or files) that can then be imported into some other Automator Installation.

To export a process

1. On the Navigation Tree, click **Automator** and select **Standard Processes**.
2. On the displayed page, select the **Export Processes** tab.
3. Select the process you want to export. Multiple selection is available.

4. Enable the **Include process dependencies** check box to export projects and project models from Server Library.
5. Specify the number of processes per file by selecting one of the following options:
 - One process per file if you want each Process to be exported to its own XML file.
 - All processes in one file if you want the exported Processes to be exported into one XML file.
6. Click the Export button.
7. In the displayed dialog box, enter the name of the output archived file.

NOTE If you selected the **One process per file**, the outputs will be named <process_name>_process.xml, located in the folder you have specified. If you selected **All processes in one file**, the output will use the name you specified, located in the folder you selected.

8. Click the **OK** button to export the process or **Cancel** to cancel exporting. A message box will inform you when the export operation has completed.

Process Handling Scenarios

When specifying general settings for a process, you must specify whether there will be one project per job or multiple projects per job for the process.

If you select the **One Project per Job** option, each project assigned to the process will spawn its own job or more than one job if that project contains multiple input files that are to be handled each in its own job. If the **Allow Parallel Jobs** option is not selected, then the job(s) from the first project will be processed before the job(s) from the second project, and so on. If allow parallel jobs is selected, however, then one cannot assume that the spawned jobs will run in any particular sequence.

If you select the multiple projects per job option on the General tab, then each job spawned will process the inputs for all the projects in their given order. Often there will be only one such job spawned, but there will be more than one job spawned if any of the projects specifies multiple input files that are to be handled each in its own job.

NOTE In the case where more than one project specifies multiple input files that are to be handled each in its own job, then the process will generate an error if the multiplicities implied by these multiple projects actually exist, and/or are not compatible with each other. See Scenario 8 further for an example of this.

Below is an analysis of the scenarios that can arise from a process containing two projects, ProjectA and ProjectB. ProjectA has input Class*.prn that expands to ClassJan.prn, ClassFeb.prn, and ClassMar.prn. ProjectB has input Employ*.prn that expands to Employ1.prn and Employ2.prn.

Scenario 1

- ProjectA set for handling multiple files "all in a single job"
- ProjectB set for handling multiple files "all in a single job"
- Processing "one project per job"

Spawns 2 jobs:

1. Job1: ProjectA with inputs ClassJan.prn, ClassFeb.prn, ClassMar.prn

2. Job2: ProjectB with inputs Employ1.prn, Employ2.prn

Scenario 2

- ProjectA set for handling multiple files "all in a single job"
- ProjectB set for handling multiple files "all in a single job"
- Processing "multiple projects per job"

Spawns 1 job:

1. Job1: ProjectA with inputs ClassJan.prn, ClassFeb.prn, ClassMar.prn AND ProjectB with inputs Employ1.prn, Employ2.prn

Scenario 3

- ProjectA set for handling multiple files "all in a single job"
- ProjectB set for handling multiple files "each in its own job"
- Processing "one project per job"

Spawns 3 jobs:

1. Job1: ProjectA with inputs ClassJan.prn, ClassFeb.prn, ClassMar.prn
2. Job2: ProjectB with input Employ1.prn
3. Job3: ProjectB with input Employ2.prn

Scenario 4

- ProjectA set for handling multiple files "all in a single job"
- ProjectB set for handling multiple files "each in its own job"
- Processing "multiple projects per job"

Spawns 2 jobs:

1. Job1: ProjectA with inputs ClassJan.prn, ClassFeb.prn, ClassMar.prn AND ProjectB with input Employ1.prn
2. Job2: ProjectA with inputs ClassJan.prn, ClassFeb.prn, ClassMar.prn AND ProjectB with input Employ2.prn

Scenario 5

- ProjectA set for handling multiple files "each in its own job"
- ProjectB set for handling multiple files "all in a single job"
- Processing "one project per job"

Spawns 4 jobs:

1. Job1: ProjectA with input ClassJan.prn
2. Job2: ProjectA with input ClassFeb.prn
3. Job3: ProjectA with input ClassMar.prn
4. Job4: ProjectB with inputs Employ1.prn, Employ2.prn

Scenario 6

- ProjectA set for handling multiple files "each in its own job"
- ProjectB set for handling multiple files "all in a single job"
- Processing "multiple projects per job"

Spawns 3 jobs:

1. Job1: ProjectA with input ClassJan.prn AND ProjectB with inputs Employ1.prn, Employ2.prn
2. Job2: ProjectA with input ClassFeb.prn AND ProjectB with inputs Employ1.prn, Employ2.prn
3. Job3: ProjectA with input ClassMar.prn AND ProjectB with inputs Employ1.prn, Employ2.prn

Scenario 7

- ProjectA set for handling multiple files "each in its own job"
- ProjectB set for handling multiple files "each in its own job"
- Processing "one project per job"

Spawns 5 jobs:

1. Job1: ProjectA with input ClassJan.prn
2. Job2: ProjectA with input ClassFeb.prn
3. Job3: ProjectA with input ClassMar.prn
4. Job4: ProjectB with input Employ1.prn
5. Job5: ProjectB with input Employ2.prn

Scenario 8.1

NOTE Scenario 8.1 is executed if the key "PumpTaskLegacyBehavior" is set to "true" in the config.xml file.

- ProjectA set for handling multiple files "each in its own job"
- ProjectB set for handling multiple files "each in its own job"
- Processing "multiple projects per job"
- Error: IncompatibleProjectInputsException

Scenario 8.2

NOTE Scenario 8.2 is executed if the key "PumpTaskLegacyBehavior" is set to "false" in the config.xml file.

- ProjectA set for handling multiple files "each in its own job"
- ProjectB set for handling multiple files "each in its own job"
- Processing "multiple projects per job"

Spawns 3 jobs:

1. Job1: ProjectA with input ClassFeb.prn AND ProjectB with input Employ1.prn
2. Job2: ProjectA with input ClassJan.prn AND ProjectB with input Employ2.prn
3. Job3: ProjectA with input ClassMar.prn

Schedules

All schedules for Standard Processes are listed on the **Schedules** page. You can use this page to [add](#), [edit](#) or [delete](#) schedules.

To add a new schedule for a Standard Process

1. On the **Schedules** page under Standard Processes, click . The Schedule page will be displayed.
2. Select the **Suspended** check box if you want to suspend the task.
3. Specify the process to which the schedule will be assigned.
4. Select the task [execution frequency](#) from the following options: **One Time, Hourly, Daily, Weekly, Monthly**.
5. Set the time you want the process to run.
6. Set the start date of the schedule.
7. Select the appropriate timezone from the drop-down list.
8. Click the **Save** button to save the schedule or **Run Now** to start the task immediately.

NOTE

If the Config service is not available and you are using Altair licensing, Automator processes will be switched to "Limp Mode." In this mode, you can run only one process (i.e., one Standard Process and one Visual Process) every hour. Moreover, you cannot run a Visual Process that involves other Standard or Visual Processes. Limp Mode is deactivated once the Config service becomes available.

To edit a schedule for a Standard Process

1. Click the name of the schedule you want to change or select the schedule by clicking anywhere on the row and then click .
2. On the displayed page, make the necessary changes and save the schedule.

For details about how to change the schedule settings, refer to [Adding a Schedule](#).

To delete a schedule for a Standard Process

1. Click  at the end of the row of the schedule you want to delete. A dialog box appears.
2. Click **Delete** to remove the schedule from the list.

Execution Frequency

Execution frequency (or run time) specifies both how often a process will run (for example, once, hourly, daily, etc.) and at what time it will run. You specify execution frequency when configuring a [schedule](#) or a [custodian schedule](#).

The options in the **Execution Frequency** pane allow you to set periodicity of launching the current process. An administrator can set the following options:

- One Time:** Select this option if you want to launch the current process only once.
 - **At:** Set the time when the current process will be launched following the HH:MM format.
 - **on (mm/dd/yyyy):** Set the date when the current process will be launched. Upon clicking the field, the [calendar](#) appears.
- Hourly:** Select this option if you want to launch the current process at the specified hour or hours.

- At... minute(s) of Every hour: Set the time when the current process will be launched following the MM format.
 - **Daily:** Select this option if you want to launch the current process in a specified number of days.
 - Every ... day(s): Set the number of days after which the current process will recur.
 - At: Set the time when the current process will be launched following the HH:MM format.
 - **Weekly:** Select this option if you want to launch the current process weekly.
 - Recur every: Set the number of weeks after which the current process will recur.
 - Select corresponding days of the week.
 - At: Set the time when the current process will be launched following the HH:MM format.
 - **Monthly:** Select this option if you want to launch the current process monthly.
 - Select the day when to perform the process.
 - ◆ If you select **Day**, set the number of days in the box.
 - ◆ If you select **Business Day**, set the business day(s) on which the current process will be launched. Business days are weekdays (i.e., Monday-Friday), unless an exception date is defined. The definition of [Exception Dates](#) can be used for holidays. When an exception day is found, the count skips to the next working day.
- The process will be executed immediately after exception date/time is over. Otherwise, the process will only run on the set schedule, with the exception of the Business Day schedule, wherein an “All day” Exception Date will be excluded in the day count.

Examples:

To run the processes on the 6th business day of the month, enter “6” in the field.

To run the processes on the 1st business day of the month, then again on the 16th to the 20th business day of the month, enter “1, 16-20” in the field.

NOTE It is possible to schedule the process on a range of working days at a given time, e.g., “5”; “1,2,22,23”; “1-3, 21-23”; “-1”; “-3--1”; and “-2-3, 5, 10.”

A negative number denotes the number of days prior to the month selected, e.g., “business day -1” of October 2019 would be September 30, 2019 (provided it is a working day). Exception Dates must skip count to the next previous working day.

- ◆ If you select **The...**, select First, Second, Third, Fourth or Last from the first drop-down list and the day of the week from the second drop-down list.
- Select the month on which to perform the action.
 - ◆ If you select the **of every n Month** option, set the number of month in the box.
 - ◆ If you select the **of selected months** option, select the appropriate check box referring to the desired month.
- In the **At** check box set the time when the current process will be launched following the HH:MM format.

Jobs

A job is the resulting task of one or more projects that comprise the associated [process](#). When a process runs, it ultimately spawns one or more jobs, with each job appearing as a row on the Automator Standard Processes **Jobs** page.

The number of jobs a process creates is determined by the following:

- ❑ Whether the one project per job or the multiple projects per job setting is selected on the [General tab](#) of the Process Properties page
- ❑ The number of input files assigned to the project on the [Monitoring tab](#).
- ❑ Whether all in a single job or each in its own job is selected in the project file.

The **Jobs** page displays the following information:

- ❑ The job's unique ID number
- ❑ The name of its associated process (January Sales)
- ❑ The status of the job, which is currently "Job completed without errors"
- ❑ The name of the server on which the job was processed
- ❑ The date and time when the job was registered
- ❑ The date and time when the job was last processed.
- ❑ The duration of job execution.

To view this page, on the **Navigation Tree**, click **Automator > Standard Processes** and select **Jobs**.

You can [delete](#), [cancel](#), and re-run jobs that have failed (due to process prerequisites that haven't been met, for example).

- ❑ To view the jobs log, click the View Job Log icon . In the **Logs** page, you can view the progress of an export (shown as a percentage) under **Project Outputs**.
- ❑ To re-run a job, click the Re-run icon .
- ❑ To reload the jobs list, click the Reload Grid icon .

NOTE

You can filter the jobs by job ID, process name, status, location, date/time registered, and date/time last processed by using the corresponding fields in the column headings.

The following filter settings are available:

~ - contains

== - equal

< - less

> - greater

X - Reset search value

Cancelling a Job

Jobs may be cancelled at a number of stages prior to their completion. For example, you may want to cancel a job that has failed to run due to a missing process prerequisite if retries are enabled for the process and you are sure that the prerequisite will not be met in the near future.

To cancel a job

1. On the Navigation Tree, click **Automator > Standard Processes** and select **Jobs**.
2. On the **Jobs** page do one of the following:
 - Select the job you want to cancel, by clicking anywhere on the row, and then click , or

- Click  at the end of the row with the job you want to cancel.

Deleting a Job

You can remove jobs from the Jobs page by deleting them. You may find it necessary to do this during testing or to simply purge old data.

NOTE Once a job has been deleted it cannot be recovered. Delete a job only if you are sure you no longer need it.

To delete a job

1. On the Navigation Tree, click **Automator > Standard Processes** and select **Jobs**.
2. On the **Data Jobs** page do one of the following:
 - Select the job you want to delete, by clicking anywhere on the row, and then click , or
 - Click  at the end of the row with the job you want to delete. A dialog box appears.

NOTE Multiple jobs selection is possible.

3. Click **Delete** to remove the process from the list.

Distributions

Distributions specify actions, called distribution actions, that distribute or handle files and information outside of Automator. Distributions can be specified for input files, export files, and portable reports, and can be configured with the Distribution Editor and the Script Editor. Distributions can be used for:

- Sending [emails](#) (with or without attachments)
- [RSS](#) feeds
- [Moving, copying](#), and [deleting](#) files (to both the file system and to FTP/SFTP-compliant and WebDAV-compliant servers using http)
- Writing to the Windows Event [Log](#)

There are two types of distributions that can be specified:

- Project Distributions:** These distributions apply only to the input, export and portable report files of the specified project.
- Global Distributions:** These distributions can apply to any input files used in all processes, and to any export files and portable reports that are produced by all processed in the specified folders.

Global distributions are useful if you have fairly narrow rules on distribution of files and information, or you want to try and structure distribution in a more transparent way, rather than having multiple discrete distribution rules in many different projects.

A typical use for this might be to set up "inboxes" and "outboxes", where files that come into the specific inboxes are renamed and copied to an archive folder and then deleted, and the projects are set up to export to a particular person's (or group's) outbox so that they receive emails or RSS feeds targeted to them.

To view the Distributions page, on the **Navigation Tree**, click **Automator > Standard Processes** and select **Distributions**.

Adding a Distribution

To create a Global Distribution

1. On the Navigation Tree, click **Automator > Standard Processes** and select **Distributions**.
2. Click the **Add** icon  to display the **Distribution** page.
3. In the **Location** box select the location type from the drop-down list.

NOTE If the required server is not on the list, refer to [Locations Management](#) topic to add a new location. In case of a single server configuration refer to the Configurator Guide in the [Monarch Server Installation Guide](#) to change location settings.

4. In the **Path** box enter the path to a file or click the **Browse** button, to select a file. The **Named Paths** window will appear.
In the displayed window do one of the following:
 - From the Named Paths list, select the appropriate [named path](#) and select the file from the list of displayed in the Named Path Content box. Click Save to add the file.
 - In the Path field, type the path to the file. Click Save to add the file.
5. Click **Add Actions** to [add actions](#) to the distribution.

Editing a Distribution

Once you have configured and saved a distribution, you can edit it at any time.

To edit a distribution

1. On the Navigation Tree, click Automator > Standard Processes and select Distributions.
2. Click the name of the distribution you want to edit or click anywhere on its row, and then click .

When you open a saved distribution for editing, the list of actions for the distribution becomes available. You can [add](#), [edit](#) or [delete](#) actions.

For details about how to change distribution parameters please refer to [Adding a Distribution](#).

Adding a Distribution Action

The specified action will be applied to any input files that are used in a process, as well as to any export files and portable reports that are produced by a process, in the folders specified.

To add an action for a distribution

1. On the **Distribution Properties** page click the **Add** icon . The Action page will display.
2. Select one of the following actions for distribution:

- [Translate](#): This distribution provides a way to translate folder names to Internet addresses. This is necessary for RSS distributions and web service actions. The Translate distribution is only available as a global distribution defined in Automator, it cannot be defined in a project.
- [Move](#): This allows files to be moved to another location, either to the file system using drive letter or UNC naming, or to FTP/SFTP and HTTP locations.
- [RSS](#): RSS (Really Simple Syndication) is a widely used format for publishing information. This is a way of allowing users to "pull" information as opposed to the "push" model of email. The RSS file must reside on a web server and be accessible to users who can read the RSS feed using one of the many free and commercial RSS news readers available on the market, for example, RSSBandit.
- [Log](#): This allows the creation of an event written to the WindowsApplication Event Log. This can be useful when using system management systems such as BMC Patrol, Tivoli, or Openview to tie in Automator operations with your overall systems management strategy. This is especially useful when combining Alerts with a log distribution.
- [Copy](#): This distribution allows files to be copied to another location, either to the file system using drive letter or UNC naming, or to FTP/SFTP and HTTP locations.
- [Email](#): To create an email distribution, you need to specify the folder (location) which will be used to detect the input and/or export files for distribution. Once specified, you can then select the sender's mail address and the "To" and "CC" addresses for the email.
- [Delete](#): This deletes all files specified by the distribution.

Translate Distribution Action

To add a translate action

1. Select the **Translate** option on the Action page.
2. In the **Url** box specify the folder for translation and the base URL. For example, you may be exporting files to a folder named z:\iispublic\automatorrss, which is accessible to the intranet at the URL <http://internal.yourcompany.com/reports/rss>.

NOTE

You may also have the scenario where you are copying the files to a different web server via a copy distribution using FTP/SFTP, and so require the RSS items to point to the files on the target web server. The translate distribution allows you to do this.

3. Click **Save** to save the action.

Move Distribution Action

To add a move action

1. Select the **Move** option on the Action page.
2. In the **Move File to Folder** box, specify the destination folder. You can either enter the destination folder path in the box or select the folder by clicking the **Browse** button. The **Named Paths** dialog box will appear.
 - a. From the **Named Paths** list, select the appropriate [named path](#).
 - b. Select the folder from the list of displayed in the **Named Path Content** box.
 - c. Click **Save** to add the folder.
3. In the **Rename file as** box, specify a new file name, if you want to change the file name.

NOTE You may use the naming macros to name the moved files and destination folder. To do this, click the **Insert Macro** icon  and select the [naming macros](#) from the drop-down list.

4. Select the **Change extension to** check box to change file's extension. Specify new extension in the enabled text box below.
5. Select the **Zip** check box, if you want to archive the file. The **Zip Options** drop-down arrow will appear.

NOTE The **Zip** check box is disabled if **Change extension to** check box selected.

6. Under **Zip Options** provide the following information:
 - Select the compression type from the following: None, Medium, Maximum.
 - Select the Store directory path check box, if you want to store directory path.
 - Select the Password check box to set the password.
7. Select the **Overwrite** check box to overwrite the existing file.
8. Click the **Authentication Credentials** drop-down arrow to specify the authentication credentials.

NOTE The **Authentication Credentials** link appears only if you enter "ftp://" or "sftp://" or "http://" in the folder path in the **Move File to Folder** box.

9. Under Authentication Credentials do the following:
 - Specify username.
 - Specify password.
 - Specify domain name.
 - Select the Ascii check box to specify an ASCII mode transfer for your FTP/SFTP distribution rather than binary mode, which is the default.

NOTE ASCII Mode should only be used for text-only files such as TXT, ASP, HTML, while binary mode can be used for anything else, for example, DOC, ZIP, XLS. If in doubt as to which transfer mode to use, go with the default mode of binary (to do so, do not select this check box).

10. Click the **Save** button to save the action.

RSS Distribution Action

To add an RSS action

1. Select the **RSS** option on the Action page.
2. In the **File** box, specify the file. You can either enter the path to a file in the box or select a file by clicking the **Browse** button. The **Named Paths** dialog box will appear.
 - a. From the **Named Paths** list, select the appropriate [named path](#).
 - b. Select the file from the list of displayed in the **Named Path Content** box.
 - c. Click **Save** to add the file.
3. Select the **Create if not found** check box, if you want to create the file if the specified not found.
4. In the **Title** box, enter the item title.
5. In the **Item description** box, enter the item description.

NOTE You may use the naming macros to provide a title and description for an RSS distribution. To do this, click the **Insert Macro** icon  and select the [naming macros](#) from the drop-down list.

6. In the **Link** box, enter the name of link to the file.

NOTE Displays the URL of the RSS item. This field is usually left blank so that Automator can automatically assign a URL (based on a translate distribution) from the input or export file that is being processed.

7. Click the **Channel** title to provide the channel parameters:
 - **Title:** Enter channel title.
 - **Channel Description:** Provide channel description.
 - **Link:** Enter the name of link to the file.

NOTE If you leave the **Title**, **Channel Description**, and **Link** fields blank the default values will be automatically assigned.

- **Max Items:** Specify the maximum number of items.
 - **Unlimited:** Select this check box if you want to set the unlimited number of items.
8. Click **Save** to save the action.

In order to use an RSS feed, you must define a global [Translate](#) distribution action. When creating an RSS feed, the location must be specified, as well as the path to the RSS file to be created or updated.

NOTE Automator uses the RSS 2.0 format when generating an RSS feed.

Log Distribution Action

To add a log action

1. Select the **Log** option on the Action page.
2. Specify the necessary information in the following fields:
 - **Type:** Select the type from the drop-down list.
 - **Event ID:** Enter the event ID.
 - **Message:** Type the message in the text field.
3. Click **Save** to save the action.

NOTE You may use the naming macros to provide a message for the log distribution. To do this, click the **Insert Macro** icon  and select the naming macros from the drop-down list.

Copy Distribution Action

To add a copy action

1. Select the **Copy** option on the Action page.
2. In the **Copy File to Folder** box, specify the destination folder. You can either enter the destination folder in the box or select the folder by clicking the **Browse** button. The **Named Paths** dialog box will appear.
 - a. From the **Named Paths** list, select the appropriate [named path](#).
 - b. Select the folder from the list of displayed in the **Named Path Content** box.
 - c. Click **Save** to add the folder.

NOTE You can also specify an FTP/SFTP site or http WebDAV repository here, for example, ftp://ftp.yourcompany.com/excelexporters.

3. In the **Rename file as** box, specify a new file name, if you want to change the file name.

NOTE You may use the naming macros to name the moved files and destination folder. To do this, click the **Insert Macro** icon  and select the [naming macros](#) from the drop-down list.

4. Select the **Change extension to** check box to change file's extension. Specify new extension in the enabled text box below.
5. Select the **Zip** check box, if you want to archive the file. The **Zip Options** drop-down arrow will appear.

NOTE The **Zip** check box is disabled if **Change extension to** check box selected.

6. Under **Zip Options** provide the following information:
 - Select the compression type from the following: None, Medium, Maximum.
 - Select the Store directory path check box, if you want to store directory path.
 - Select the Password check box to set the password.
7. Select the **Overwrite** check box to overwrite the existing file.
8. Click the **Authentication Credentials** drop-down arrow to specify the authentication credentials.

NOTE The **Authentication Credentials** link appears only if you enter "ftp://" or "sftp://" or "http://" in the folder path in the **Copy File to Folder** box.

9. Under Authentication Credentials do the following:
 - Specify username.
 - Specify password.
 - Specify domain name.
 - Select the Ascii check box to specify an ASCII mode transfer for your FTP/SFTP distribution rather than binary mode, which is the default.

NOTE ASCII Mode should only be used for text-only files such as TXT, ASP, HTML, while binary mode can be used for anything else, for example, DOC, ZIP, XLS. If in doubt as to which transfer mode to use, go with the default mode of binary (to do so, do not select this check box).

10. Click the **Save** button to save the action.

Email Distribution Action

To add an email action

1. Select the **Email** option on the Action page.
2. Specify the recipients' email addresses in the corresponding boxes: **From, To, Cc, Bcc**. To select recipients from the [contacts list](#), click the **Contacts** icon . The **Find Contacts** window will appear. In the **Find Contacts** window:
 - a. Select the **Look In** criteria from the following:
 - ◆ **Internal**: Select from contacts specified in the Automator Contacts.
 - ◆ **Active Directory**: Selects from contacts specified in the active directory.
 - ◆ **Domino NOTES**: Select from contacts specified in Domino NOTES.
 - b. Select the Search Criteria from the following options: Name, Email, Contains, Is, Starts with, Ends with.

- c. Click the **Find Now** button, to display all the available contacts according to the specified criteria.
 - d. Select the necessary contacts from the Available Contacts list.
 - e. Click **Save** to insert specified recipients.
3. Type the message subject in the **Subject** box.

NOTE You may use the naming macros to provide a subject line for an email distribution. To do this, click the **Insert Macro** icon  and select the naming macros from the drop-down list.

4. Type the message in the text box.
5. Select the **Attach files** check box to attach a file for the distribution.
6. Select the **Zip** check box to archive the attached files. The **Zip Options** drop-down arrow will appear.
7. Under **Zip Options** provide the following information:
 - Specify the compression type from the following: None, Medium, Maximum.
 - Select the **Store directory path** check box if necessary.
 - Select the **Password** check box to set the password.
8. Select the **Attach links** check box to attach a link to file.

NOTE Please specify the **Admin Host and Port** and **Client Host and Port** information in the [System Settings](#).

9. Select the **One file per mail** check box to attach only one file per mail.
10. Select the email priority from the following: **Low**, **Normal**, **High**.
11. Click **Save** to save the action.

Delete Distribution Action

To add a delete action

1. Select the **Delete** option on the **Action** page.
2. Click **Save** to save the action.

Editing a Distribution Action

Once you have configured and saved a distribution action, you can edit it at any time.

To edit a distribution action

1. On the Navigation Tree, click Automator > Standard Processes and select Distributions.
2. Click the name of the distribution you want to edit or click anywhere on its row, and then click .

3. Click the name of the action you want to edit or click anywhere on its row, and then click  .
4. Make necessary changes to the selected action. For details about how to change action settings please refer to the corresponding topic:
 - [Translate Action](#)
 - [Move Action](#)
 - [RSS Action](#)
 - [Log Action](#)
 - [Copy Action](#)
 - [Email Action](#)
 - [Delete Action](#)

Deleting a Distribution Action

You can delete any action on the **Action** page.

To delete an action

1. On the Navigation Tree, click Automator > Standard Processes and select Distributions.
2. Click the name of the distribution you want to change or click anywhere on the row, and then click  .
3. On the **Distribution Properties** page, click  at the end of the row with the action you want to delete. A dialog box appears.
4. Click **Delete** to remove the action from the list.

Deleting a Distribution

You can delete any distribution on the **Distributions** page.

NOTE When you delete a distribution, all of its associated actions are also deleted.

To delete a distribution

1. On the Navigation Tree, click Automator > Standard Processes and select Distributions.
2. Click  at the end of the row with the distribution you want to delete. A dialog box appears.
3. Click **Delete** to remove the distribution from the list.

Alerts

Automator provides an alerting mechanism whereby users can be automatically notified of certain conditions. For example, you can define an alert that will notify you whenever a process fails to run.

Automator includes five predefined alerts that you can use, and any number of additional alerts can be defined using scripting.

All the existing in the system alerts are displayed on the **Alerts** page. To view the **Alerts** page, click **Automator > Standard Processes** and select **Alerts**.

Use this page to [add](#), [edit](#), or [delete](#) alerts.

Using a Script to Define an Alert

Scripts can be used to define alerts. For example, suppose the PostExport code examines the outputs listed in the job log and determines that something is amiss. It could then raise an alert as follows:

```
Log.AddEvent ("The record count for the balance is outside specified bounds", "BalanceProcessAlert")
```

This would raise an alert named "BalanceProcessAlert".

This alert could then be assigned an email distribution action to notify a technician of the problem via an email message.

Adding an Alert

Automator provides an alerting mechanism whereby you can be notified based upon certain conditions. You can, for example, be notified via email if a process fails to run. These notifications are called alerts. Some alerts are pre-defined within Automator itself, such as the process failure (SystemAlertProcessFailed) and job failure (SystemAlertJobFailed) alerts, while others can be defined by users in global, pre-export or post-export scripts, using the AddEvent method from the JobLog object.

To add an alert

1. On the **Navigation Tree**, click **Automator > Standard Processes** and select **Alerts**. The Alerts page displays.
2. On the Alerts page, click the **Add** icon  to create a new alert.
3. In the **Code** box specify the alert type.

NOTE You may type a new alert name or select one of the following predefined alert types:

- SystemAlertProcessFailed**: Indicates that a [process](#) has failed.
- SystemAlertJobFailed**: Indicates that a [job](#) has failed.
- SystemAlertNoInputToExporter**: Indicates that there was no input to the Exporter (which means, that no [input file](#) exists).
- SystemAlertDistributionError**: Indicates that a [distribution](#) has failed.
- SystemAlertVerifyFailed**: Indicates a project verification failure.

If a predefined alert (for example, the SystemAlertJobFailed) has already been selected, it will no longer be listed in the Code drop-down list, as you cannot specify two predefined alerts of the same type.

4. In the **Description** field enter alert description.
5. Click the **Ignore Global Alert** check box to ignore global alerts.

NOTE This option is available only when the **Allow ignore Global alerts** setting is enabled. See the [General Settings](#) in [Automator Settings](#) for details.

6. Click **Add Actions** to add actions to the alert.

NOTE When you add an alert, you must specify a distribution action for it. The distribution action you assign to the alert determines how you are notified of the problem. See [Alert Distribution Actions](#) below for more information.

Editing an Alert

Once you have configured and saved an alert, you can edit it at any time.

To edit an alert

1. On the Navigation Tree, click **Automator > Standard Processes** and select **Alerts**.
2. Click the name of the alert you want to change or click anywhere on its row, and then click .

When you open a saved alert for editing, the list of actions for the alert becomes available. You can [add](#), [edit](#) or [delete](#) actions.

For details about how to change alert settings please refer to [Adding an Alert](#).

Adding an Alert Action

When adding an alert, you must specify a distribution action for it. The distribution action you assign to the alert determines how you are notified of the problem. The Alert Properties page displays the list of actions that can be specified for the alert.

To add an action for an alert

1. On the **Alert Properties** page click the **Add** icon . The Action page will display.
2. Select one of the following actions for alerts:
 - [Email](#): Select to send an email notification. Note that if the email connection is down at the time the alert is invoked, the alert will also be written to the Windows Application Event Log.
 - [RSS](#): Select to specify an RSS notification.
 - [Log](#): Select to specify an Application Event Log notification. This can be useful if you use network management software such as BMC Patrol, IBM Tivoli, HP Openview or other software, to provide tighter integration into your systems management.

NOTE If an alert already has an action set, you must add another action if one is required, rather than creating a new alert.

Email Alert Action

1. Select the **Email** option on the **Action** page.
2. Specify the recipients' email addresses in the corresponding boxes: **From**, **To**, **Cc**, **Bcc**. To select recipients from the [contacts list](#), click the **Contacts** icon . The **Find Contacts** window will appear. In the **Find Contacts** window:

- a. Select the **Look In** criteria from the following:
 - ◆ **Internal**: Select from contacts specified in the Automator Contacts.
 - ◆ **Active Directory**: Selects from contacts specified in the active directory.
 - ◆ **Domino NOTES**: Select from contacts specified in Domino NOTES.
 - b. Select the Search Criteria from the following options: Name, Email, Contains, Is, Starts with, Ends with.
 - c. Click the **Find Now** button, to display all the available contacts according to the specified criteria.
 - d. Select the necessary contacts from the Available Contacts list.
 - e. Click **Save** to insert specified recipients.
3. Type the message subject in the **Subject** box.

NOTE You may use the [naming macros](#) to provide a subject line for an email distribution. To do this, click the **Insert Macro** icon  and select the naming macros from the drop-down list .

4. Type the message in the text box.
5. Select the **Attach files** check box to attach a file for the alert.
6. Select the **Zip** check box to archive the attached files. The **Zip Options** drop-down arrow will appear.
7. Under **Zip Options** provide the following information:
 - Specify the compression type from the following: None, Medium, Maximum.
 - Select the **Store directory path** check box if necessary.
 - Select the **Password** check box to set the password.
8. Select the **Attach links** check box to attach a link to file.

NOTE Please specify the **Admin Host and Port** and **Client Host and Port** information in the [System Settings](#).

9. Select the **One file per mail** check box to attach only one file per mail.
10. Select the email priority from the following: **Low**, **Normal**, **High**.
11. Click **Save** to save the action.

RSS Alert Action

To add an RSS action

1. Select the **RSS** option on the Action page.
2. In the **File** box, specify the file. You can either enter the path to a file in the box or select a file by clicking the **Browse** button. The **Named Paths** dialog box will appear.
 - a. From the **Named Paths** list, select the appropriate [named path](#).
 - b. Select the file from the list of displayed in the **Named Path Content** box.

- c. Click **Save** to add the file.
3. Select the **Create if not found** check box in order to create the file, if the specified not found.
4. In the **Title** box, enter the item title.
5. In the **Item description** box, enter item description.

NOTE You may use the [naming macros](#) to provide a title and description for an RSS distribution. To do this, click the **Insert Macro** icon  and select the naming macros from the drop-down list.

6. In the **Link** box, enter the link name.

NOTE Displays the URL of the RSS item. This field is usually left blank so that Automator can automatically assign a URL (based on a [translate distribution](#)) from the input or export file that is being processed.

7. Click the **Channel** title to specify the channel parameters:
 - **Title:** Enter channel title.
 - **Channel Description:** Provide channel description.
 - **Link:** Enter link name.

NOTE If you leave the **Title**, **Channel Description**, and **Link** fields blank the default values will be automatically assigned.

- **Max Items:** Specify the maximum number of items.
 - **Unlimited:** Select this check box if you want to set the unlimited number of items.
8. Click **Save** to save the action.

In order to use an RSS feed, you must define a global [Translate distribution action](#). When creating an RSS feed, the location must be specified, as well as the path to the RSS file to be created or updated.

NOTE Automator uses the RSS 2.0 format when generating an RSS feed.

Log Alert Action

To add a log action

1. Select the **Log** option on the **Action** page.
2. In the **Type** box, select the type from the drop-down list.
3. In the **Event ID** box, enter the event ID.

4. In the **Message** text field, type the message.

NOTE You may use the naming macros to provide a message for the log alert. To do this, click the **Insert Macro** icon  and select the naming macros from the drop-down list.

5. Click **Save** to save the action.

Editing an Alert Action

Once you have configured and saved an alert action, you can edit it at any time.

To edit an alert action

1. On the Navigation Tree, click **Automator > Standard Processes** and select **Alerts**.
2. Click the name of the alert you want to change or click anywhere on its row, and then click .
3. From the action list select the one you want to edit. For this do one of the following:
 - Click the name of the action you want to change or click anywhere on its row, and then click .
4. Make necessary changes to the selected action.
5. To save the changes, click **Save** to save the action.

For details about how to change action settings:

- [RSS Action](#)
- [Log Action](#)
- [Email Action](#)

Deleting an Alert Action

You can delete any action for the alert on the **Action** page.

To delete an action

1. On the Navigation Tree, click **Automator > Standard Processes** and select **Alerts**.
2. Click the name of the alert you want to change or click anywhere on its row, and then click .
3. Click  at the end of the row with the action you want to delete. A dialog box appears.
4. Click Delete to remove the action from the list.

Deleting an Alert

You can delete any alert on the **Alerts** page.

NOTE When you delete an alert, all of its associated actions are also deleted.

To delete a distribution

1. On the Navigation Tree, click **Automator > Standard Processes** and select **Alerts**.
2. Click **X** at the end of the row with the alert you want to delete. A dialog box appears.
3. Click **Delete** to remove the alert from the list.

Global Environment Scripts

Global environment scripts allow you to enter functions, classes, variables, etc. that are available for, or will execute over, all processes. They also allow the setting of global assembly imports and references, which are valid over Automator process boundaries. Global environment scripts are split into three sections:

- [Global Declarations](#): Used for variables, classes, functions, subroutines, etc., that need to be available across all Automator processes.
- [References and Imports](#): Used for importing .NET namespaces and .NET assemblies for global use.
- [Passwords](#): Used to specify passwords for use in password macros.

For global environment scripting, access the Script Editor by clicking **Automator > Standard Processes** on the **Navigation Tree** and then selecting **Scripts** from menu.

Project Editor

Project files are special files created with Project Editor (or with Monarch) that have a .xprj or .dprj extension and which contain all of the information necessary to execute one or more Automator operations. Projects utilize text or database input, extract data from the input via Monarch model files, and export the extracted data to various file types (for e, XLS, PDF, MDB, TXT, HTML, XML, etc.).

Project files may contain:

- Information about the project's data source (e.g., either one or more report files or an external database)
- The path and filename of the model to be applied to the data source
- Data extraction templates and field properties
- Filters, sort orders, and summary information
- Calculated field definitions
- Information about project exports, portable report settings, and process prerequisites

Projects are a key component of processes as when you [create a process in Automator](#), one of the key steps you perform is to assign a project (or projects) to it.

With Automator you can use an existing project file created with Monarch or you can create a new project file via the Project Editor. For details refer to topic [Creating Project Files](#).

Creating Project Files

When creating a new project file via the Project Editor, you may perform the following steps:

- [Specify general settings](#) — General settings specify an author for the project file, its name and provide a description of it.
- [Specify a model setting](#) — When you specify a model setting, you assign a model to the project, and specify the desired verification settings, if any.
- [Specify input settings](#) — Input settings specify the input file type (i.e., text or database), the input file(s) you want to assign to the project, how multiple input files should be grouped, etc.

- ❑ [Specify export settings](#) — When you specify project export settings, you specify an export (or exports) of export files and specify any distribution actions you want to apply to them.
- ❑ [Specify prerequisite settings](#) — When you specify prerequisite settings, you add a file prerequisite or a command line prerequisite to the project. Any prerequisites you add must first be met if the associated process is to run.
- ❑ [Specify script settings](#) — Script settings allow you to, among other things, specify pre-export and post-export scripts to modify the input and export files.

NOTE Depending on your needs, not all of these steps may have to be performed. Specifying [general](#) and [model](#) settings is enough to create a project.

Specifying General Settings for a Project File

When you specify general settings for a project, you specify an author for the project file, its name and provide a description of it.

To specify general settings for a project

1. On the **Navigation Tree**, click **Automator > Standard Processes** and select **Project Editor**. The Home tab is displayed.
2. In the **Author** field, enter the desired author's name.
3. In the **Name** field, enter project name.
4. In the **Description** field, enter a description of the project.
5. Proceed to the **Model and Verifications** tab to specify model settings for the project.

Viewing Project Information

Under **Project Information** heading you can view, copy, and print the information contained in the current project. It also provides access to the Audit Trail.

NOTE The contents of the Project window vary, as they are determined by the contents of the current project.

The Project Information section displays information such as:

- ❑ **Author:** Displays the author of the current project.
- ❑ **Description:** Displays the current description of the project file.
- ❑ **Current State:** Displays the currently active view, and the current summary, filter or sort (if any).
- ❑ **Model:** Displays the path of the project's model file.
- ❑ **Report:** Displays the path of the project's report file.
- ❑ **Project Exports:** Displays information about the project's export(s), such as the name and export location of the export file, whether a filter is assigned to the table, etc.
- ❑ **Audit Trail:** Displays the audit trail for the current project.

To print the information contained in the current project, use the **Print** icon in the upper-right corner of the Project Information section.

Specifying Model Settings for a Project File

When you specify model settings for a project, you select the model you want to assign to it, and specify any desired verification settings.

To specify model settings for a project

1. On the Navigation Tree, click Automator > Standard Processes and select Project Editor.
2. On the **Model and Verifications** tab, click the **Browse** button next to the **Location** field, to select the desired model.
3. In the displayed window, select the data source type from the following: Local, Named Paths, Web, or Server Library.
4. Depending on the data source type specify other parameters:
 - **Local:** Specify project location and path to it.
 - **Named Paths:** Select the appropriate [named path](#) and select the project file path from the list of displayed in the Named Path Content box.
 - **Web:** Specify project location.
 - **Server Library:** Select server library type and select the server library from the list of displayed in the Available Server Library box

NOTE Model files in compressed archive format (e.g., zip and gzip) can be used and specified in the Project Editor.

5. If you want to specify verification settings for report inputs, select the desired options under the **Model Verifications** heading.

NOTE Verification is not available for database inputs. See [Verifying Projects](#) for additional information.

6. To save or download created project file, click the corresponding button or continue specifying other project settings if necessary.

NOTE When saving a project file provide a comment in the displayed window and click **Commit**. The Server Library [Project page](#) is displayed, where you can edit project file name, revision and access rights if necessary.

The next step in creating a project is to [specify input settings](#) for it.

Verifying Project Reports

Automator supports verification of report inputs on a project-by-project basis.

NOTE Verification is not available for database inputs. If a project calls for database inputs, the verification options are simply ignored.

Verification settings allow you to specify the kind of verifications you want it to perform when processing the project's associated job. When project verification is enabled, Automator may verify field boundaries, data types and filter results.

NOTE Automator performs verifications after running the PreExport script (if any), and before performing any exports.

Specify the desired verification settings by selecting the appropriate check boxes on the **Model and Verifications** tab. The following types of verification are available:

- Following filter produces at least one row** — Checks that when the specified filter is applied to the data, the resulting table contains at least one row.
- Following filter produces no rows** — Checks that when the specified filter is applied to the table, no rows are produced. For example, you may have created a filter that performs checks on your data to ensure that it is valid, and if any bad rows are returned by this filter, you wish to abort the procedure.
- Field boundaries and data types for text inputs** — Checks that all fields are in the correct location, are the correct length, and are of the correct data type. This is equivalent to using the verify feature in Monarch. Note that any verification settings recorded in the Monarch model will be honored. These can be found in the Field List of Monarch.

NOTE This feature is not available for database and PDF inputs.

- Unfiltered table has at least one row** — Checks that the unfiltered table contains at least one row of data.
- Text field lengths** — Checks that all text fields have valid field lengths.
- Numeric field decimals** — Checks that all numeric fields have valid decimals.
- Numeric field lengths** - Checks that all numeric fields have valid field lengths.
- Database fields** — Checks that all database fields have been defined correctly.
- Calculated fields** — Checks that all calculated fields have been defined correctly.
- Date/time format precisions** — Checks that all date/time fields have the correct format precision.

When you select a verification option, project verification is activated. Note that the default **If verification fails** setting is **Continue**. This means Automator will continue processing the associated job even if a verification failure of the specified kind is encountered. If you prefer to have Automator stop the job when a verification failure occurs, select the **Stop** option instead. If you want to stop the job without sending an email alert, select the **Stop without error notifications** option.

The **If verification fails** option applies to the entire set of verifications. If this option is **Stop**, then the first failed verification causes the job to immediately fail, thereby skipping any subsequent verifications. If this option is **Continue**, then all the verifications are performed regardless of their success or failure. In either case, the verification results are noted in the **<verification>** section of the job log.

If verification fails, a "SystemAlertVerifyFailed" alert is raised in the job log, then the job either continues or fails, according to the specified continue/stop option.

Whenever Automator performs verification, it writes a **<verification>** section to the job log, regardless of whether verification is successful or not. The verification section of the job log looks similar to this when successful:

```

<verification>
  <verify_project id="1">
    <field_properties>
      <success>
        <model>
          <location>C:\Test\Models\V7\Lesson11.xmod</location>
        </model>
      </success>
    </field_properties>
  </verify_project>
</verification>

```

If a verification failure occurs, the **<verification>** section of the job log will contain some details about the type of failure and its specific location within the input report(s).

Viewing Model Information

Model Information section provides detailed information about the Monarch model assigned to the current project. Here you can view, copy and print the information contained in the current model. It also provides access to the Audit Trail.

NOTE The contents of the Model window vary, as they are determined by the contents of the current model.

The **Model Information** section displays information such as:

- ❑ **Author:** Displays the author of the current model.
- ❑ **Description:** Displays the description of the current model.
- ❑ **Input Settings:** Displays information about the current input settings, such as whether to ignore leading control characters and form feeds, whether to interpret input as ASCII, when to force a page break (i.e., after n lines), etc.
- ❑ **Templates:** Lists the templates within the model, provides information about them (e.g., their names, whether they are floating traps or multi-column traps, their trap characters, etc), and displays sample lines of the data they trap.
- ❑ **Fields:** Lists the fields within the model and provides information about them, e.g., their names, their type (character or numeric), their data and display lengths, their alignment, etc.
- ❑ **Filters:** Lists filters defined for the model. Each entry on the list shows the name of the filter, the filter type, and the filter expression.
- ❑ **Sorts:** Lists sorts defined for the model. Each entry on the list shows the sort name, sort fields used, and sort order (ascending or descending).
- ❑ **Summaries:** Lists summaries defined for the model. Each entry on the list shows the summary name, filter used, custom page setup, number of keys, and number of measures.
- ❑ **Key Fields and Measures:** Enumerates key fields and measures for each summary in the Summaries list. For each Key Field, the field names, titles, display lengths, and sort directions are displayed. For each Measure, the measure title, display length, and expression are displayed.
- ❑ **Audit Trail:** Displays the audit trail for the current model.

Specifying Input Settings for a Project File

You can specify the following the input settings for a project file:

- The input file type: either text or database
- The input file(s) to be assigned to the project
- How multiple input files should be grouped
- The desired file existence criteria
- The order in which multiple input files will be processed
- An input file distribution, with which you can email, copy, move, etc, the input file(s).

To specify input settings for a project

1. On the Navigation Tree, click Automator > Standard Processes and select Project Editor.
2. On the **Input** tab specify the input file type by selecting either the **Text** or **Database** radio button. If you select **Database**, perform the following steps, then skip down to [Step 5](#):
 - Under the Select a data source heading, specify the data source or click the Browse For... button to select the data source type (Local or Web File, Data Link Wizard, Data Link File or Connection Definitions) and the desired file via the dialog that displays.

NOTE In addition to Excel file types .xls and .xlsx, .xlsb and .xlsm file types may be used as database file input.

- If the file is password protected, select the Include password check box and enter the required password in the adjacent field.
- If you want to select a specific table within the input file to export data to, click the Table drop-down arrow and select the desired table.
- If you want to replace nulls in the table, select the Replace nulls with default values (empty strings, zeros, and default dates (1/1/1970)) check box.
- To apply a filter (or filters) to the table during the export select the Apply filter check box and click the Change button under it to specify necessary filters.

NOTE When Automator exports the table data, the filter(s) you selected will be applied to it. Only the data that matches the filter's specifications will be exported.

- To include column names in the table with the exported data, select the First row contains column names check box.
- To choose an Excel engine, click on the **Excel Engine** drop-down list, and choose **Legacy** or **Modern**.

NOTE For the Modern Excel engine, password-protected Excel files can be used.

3. If you select **Text**, click the **Add** button to add an input file to the project via the displayed dialog. Repeat this step to add any additional input files to the project.

NOTE Input files in compressed archive format (e.g., zip and gzip) can be used and specified in the Project Editor.

4. If you added more than one input file to the project, specify how you want to group the multiple files by selecting the **All in a single job** or the **Each in its own job** radio button.

NOTE This provides a fundamental choice as to how jobs are spawned when the process is assigned more than one project file. The option makes no difference if the process specifies only a single project. For more information, see [Process Handling Scenarios](#).

5. Specify a **File Existence Criteria** setting by selecting one of the following options:
 - **All specified files must exist:** Specifies that all of the input files must exist for the process to start.
 - **At least one must exist:** Specifies that at least one of the input files must exist for the process to start.
 - **Not required:** Specifies that none of the input files must exist for the process to start.
6. Specify an **Order in which to process multiple files** setting by selecting one of the following options:
 - **Natural:** Select to process the input files in the order in which they are listed (i.e., from top to bottom of the input file list).
 - **Alphabetical:** Select to process the input files alphabetically according to their names.
 - **Oldest first:** Select to process the input files in chronological order, i.e., from oldest to newest.
 - **Newest first:** Select to process the input files in reverse chronological order, i.e., from newest to oldest.
7. To save or download created project, click the corresponding button or continue specifying other project settings if necessary.

NOTE When saving a project file provide a comment in the displayed window and click **Commit**. The Server Library [Project page](#) is displayed, where you can edit project file name, revision and access rights if necessary.

You may now perform the next step in configuring a project file, [specifying export settings](#).

If you want to add an input distribution to the project, navigate to the **Input Distribution** tab and follow the instructions provided in the [Distributions](#) section.

If in Monarch Server License the **Enterprise Server** component enabled, you can define document types from the Report Warehouse as input for a project.

To specify Report Warehouse as input settings for a project

1. On the Navigation Tree, click Automator > Standard Processes and select Project Editor.
2. On the **Input** tab, select the **Text** radio button to specify the input file type and click the **Add** button to add the input file to the project via the displayed **Set Report File** dialog.

3. In the **Set Report File** dialog, select **Report Warehouse** from the drop-down list in the **Source Type** field.
4. Specify document type of the reports to be considered as input. Click **Select** and in the displayed **Select document type** dialog select the appropriate item in the list of existing document types.
5. Specify time for launching the process. You can select the following options:
 - **When the specified number of files appear:** specify number of new reports appeared after which the process will be launched.
 - **By date:** a range of new reports (based on filing or document date) for which the process will be launched. Use calendar to specify the date.
 - **For newest reports:** the process will be launched as soon as any new report appears.

Specifying Export Settings for a Project File

Using Automator Export Wizard, you can configure project exports to export table or summary data to a file or database. When the project's associated process is run, the export(s) will be performed. When you specify export settings for a project, you specify an export (or exports) of files as well as any [distribution actions](#) you want to apply to it.

NOTE You can also specify default export file types at this time.

To specify export settings for a project

1. On the Navigation Tree, click Automator > Standard Processes and select Project Editor.
2. On the **Exports** tab click the **Add** icon  to display the Export Wizard.
3. Step through the screens of the Export Wizard, specifying the desired settings as you go. The Export Wizard is comprised of the following screens:
 - [General screen](#)
 - [Table View screen](#)
 - [Summary View screen](#)
 - [Output File screen](#)
 - [PDF Security screen](#)
 - [PRF Info screen](#)
 - [Tableau Server screen](#)
 - [OleDb Info screen](#)
 - [Table Info screen](#)
 - [Column Info screen](#)
 - Select Columns to Map dialog
 - [Excel Advanced Features screen](#)

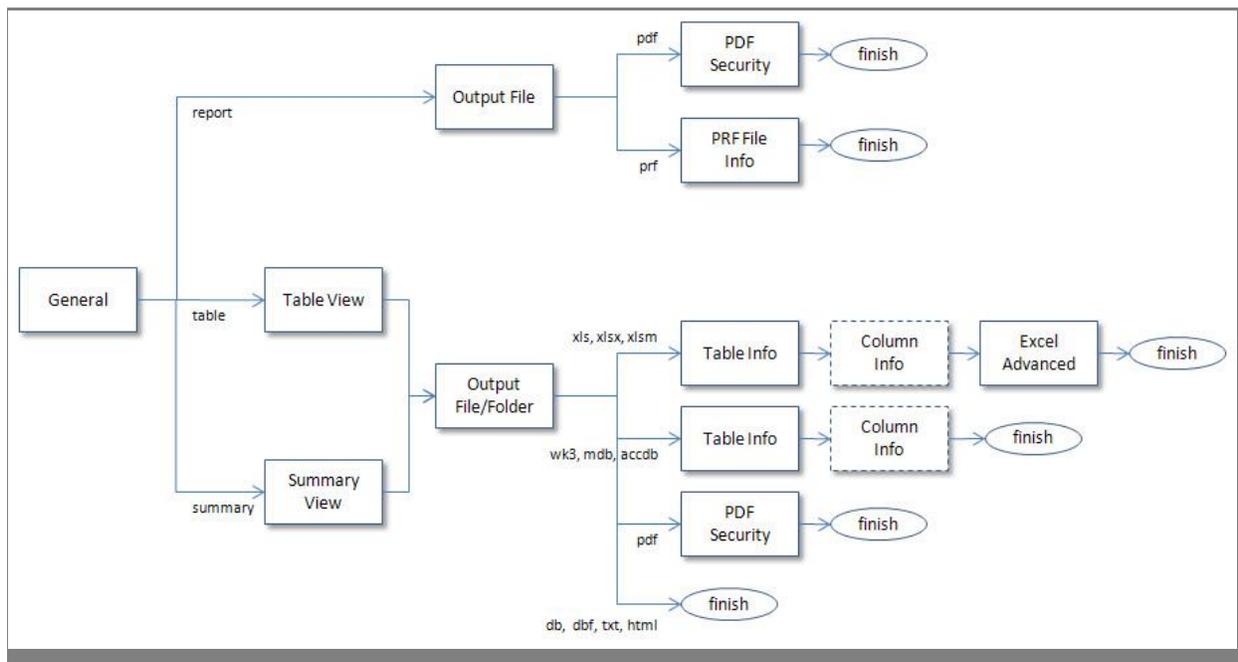
NOTE A set of screens you will traverse depends on your settings. Refer to the export wizard workflow below.

4. On the final screen of the wizard, click the **Finish** button to save the export settings and close the wizard.

5. To [specify the default version](#) for each supported export file type when exporting table or summary data for the current project, click the **Types...** button.
6. To save or download created project, click the corresponding button or continue specifying other project settings if necessary.

NOTE When saving a project file provide a comment in the displayed window and click **Commit**. The Server Library [Project page](#) is displayed, where you can edit project file name, revision and access rights if necessary.

You may now perform the next step in configuring a project file, [specifying export settings](#).



General Screen

The General Screen is part of the Export Wizard. Use this screen to:

- Name the export
- Specify whether you are exporting the report, table or summary

The **General** screen includes the following settings:

- Name:** Names the export.
- Report:** Exports to a report (text, PRF, or PDF).
- Table:** Exports table data.
- Summary:** Exports summary data.
- Comment:** Displays the comments about the export. These comments may help to remind you of important facts about the export.
- Next:** Displays the [Output File](#), [Table View](#) or [Summary View](#) screen depending on your settings (for details, see [Export Wizard workflow](#)).

- Cancel:** Cancels the export operation and closes the Export Wizard.

Table View Screen

The **Table View** screen is part of the Export Wizard. When configuring a table export, use this screen to select a filter and/or sort order to be applied to the table data. This screen is displayed when you click the **Next** button on the [General screen](#) of the Export wizard.

Dialog box options:

- Description:** Displays a description of the current export (e.g., "Export the table data").
- Filtering:** Select the filter you want to apply to the table or summary or select the "All filters" option to apply all of the available filters or the "No filter" option to apply none of them. If you choose the All filters option, you also need to specify an Automatic Naming option:
 - By files produces a file for each filter (this is the default)
 - By tables produces a table for each filter
- Sorting:** To apply a sort order to the data you are exporting, select the desired sort order from the drop-down list.
- Output using:** Select how the output will be produced:
 - File System
 - Oledb provider
- Back:** Displays the General screen of the Export Wizard.
- Next:** Displays the [Output File](#) screen (the Output Folder screen when exporting all filters and an Automatic Naming setting of "by files" is selected) or the [OleDb Info](#) screen (if OleDb provider is selected).
- Cancel:** Cancels any changes you have made and closes the Export Wizard.

To view the Export Wizard workflow, go to [Specifying Project Export Settings](#).

Output File Screen

The **Output File** screen is part of the Export Wizard. When configuring a table or summary export, use this screen to specify the folder you want to export files to.

Dialog box options:

- Save as:** Enter the path to the desired export file or click the Browse button and use the dialog that displays to select the export location. This option is available when exporting all filters and an Automatic Naming setting of "by tables" is selected.

NOTE Specifying a .PRF or .PDF extension will generate the corresponding file type accordingly. Specifying .TXT or any other extension will generate a file with text-based content only.

- Save to:** Enter the path to the desired export folder or click the Browse button and use the dialog that displays to select the export location.
- File Naming:** Enter the name of the exported file.

NOTE You may also use the naming macros to name the exported files and destination folder. To do this, click the **Insert Macro** icon  and select the [naming macros](#) from the drop-down list.

- Extension:** Select the file extension of the desired export file format.

NOTE The **Save to**, **File Naming** and **Extension** options are available when exporting all filters and an **Automatic Naming** setting of "by files" is selected.

- When output files exist:** If a file of the specified name and type already exists in the export location, select one of the following options:
 - **Overwrite file:** Overwrites the data of the existing output file with the data in the current export file.
 - **Add data to file:** Adds the export file data to the end of the specified file.
 - **Skip file:** Used for Automator project exports, this allows you to create a project export which will skip the export operation if a file of the same name already exists. This is useful for automated operations, where you may not want to overwrite a file that already exists.
- Format for Text File(s):** Allows you to specify the export format for text files. The format you select will be saved in the project file:
 - **Fixed-length:** Select to specify "fixed-length" as the export format for text files.
 - **Delimited:** Select to specify "delimited" as the export format for text files.

NOTE When creating a new export job or modifying a non-text export job, the default text export format is fixed-length for ".txt" extensions and delimited for other extensions. This assures backward compatibility with old projects.

- Cancel:** Cancels any changes to the settings and closes the Export Wizard.
- Back:** Displays the [Table View](#) or [Summary View](#) screen.
- Next:** Displays the [Table Info](#), [Tableau Server](#) or the [PDF Security](#) (if .pdf format is selected) screen.
- Finish:** Performs the table or summary export and closes the Export Wizard (if .db, .dbf, .txt, .qvx, .html format is selected).

Summary View Screen

The **Summary View** screen is part of Export Wizard, which you can use to export report, table or summary data to a file, multiple tables within a file, or multiple files in a folder. The Summary View screen displays when, on the Export Wizard's General screen, you select the **Summary** option and then click the **Next** button.

Dialog box options:

- Summary:** Select the summary that you want to export or select the "All summaries" option to export all of the available summaries.
- Automatic Naming:** Select one of the following options to specify how you want Automator to name the export:

- **By files:** Specifies that, when exporting multiple summaries, a file is exported for each summary, and the files are named according to the summary names.
 - **By tables:** Specifies that, when exporting multiple summaries, a table is produced in the export file for each summary.
- Drill State:** Select one of the following options to specify the desired drill state:
- **Most detailed:** Select to export the summary in the most detailed (i.e., drilled down) drill state.
 - **Current:** Select to export the summary in the current drill state.
- Measure:** When exporting an across summary that contains multiple measures, select one of the following options:
- **Active measure or measure grouping:** Select to export only the current measure or measure grouping.
 - **All measures as separate exports:** Select to export all of the measures as separate exports. **NOTE:** If you choose this option, specify the desired Automatic Naming setting, either "by files" or "by tables".
- Split:** Select the Each distinct value of the first key makes a separate file or table check box to have Automator split the summary export according to each value change in the first key (i.e., the leftmost one).

NOTE If you choose this option, specify the desired Automatic Naming setting, either "by files" to have each distinct value of the first key make a separate file, or "by tables" to have each change of it make a separate table.

- Filtering:** Select the filter you want to apply to the table or summary or select the "All filters" option to apply all of the available filters or the "No filter" option to apply none of them.

NOTE If you choose the "All filters" option, specify the desired Automatic Naming setting, either "by files" or "by tables".

- Sorting:** Select the sort you want to apply to the table or summary or select the "No Sort" option to apply none of them.
- Output Using:** Select how the output will be produced:
- File System
 - Oledb provider
- Next:** Displays the [Output File](#) (the Output Folder screen when exporting all filters and an Automatic Naming setting of "by files" is selected) or the [OleDb Info](#) screen (if OleDb provider is selected).
- Cancel:** Cancels the export operation and closes the Export wizard.

To view the Export Wizard workflow, go to [Specifying Project Export Settings](#).

PDF Security Screen

This screen displays when, during the configuration of a project export, you select PDF from the **Extension** drop-down list on the [Output File](#) screen of the Export Wizard, then click the **Next** button. Use this screen to specify security and encryption permissions when exporting to a PDF file.

The **PDF Security** screen includes the following options:

- Description:** Displays a description of what the export will do when run.
- Open Password options:

- **Apply:** Password-protects the PDF file.
 - **Change Open Password:** Displays the Change PDF Export Open Password dialog. Use this dialog to specify the password that must be supplied to open the PDF file.
- Permissions options:**
These options let you restrict the permissions available to users of the PDF file.
- **Apply:** Specifies permission options for the PDF file.
 - **Change Permissions Password:** Displays the Change PDF Export Permission Password dialog, with which you can specify a password for the exported PDF file(s).
 - **Printing allowed:** Allows the PDF file to be printed.
 - **Content extraction allowed:** Allows copying (e.g., cut and paste) of the PDF file's content.
 - **Editing allowed:** Allows editing of the PDF file.
 - **Annotation (comments) allowed:** Allows comments to be added to the PDF file.

NOTE If you do not specify PDF file permissions, users will have full access to it.

- Encryption options:**
Select one of the following encryption options.

NOTE One (or both) of the passwords must be applied to enable encryption.

- **RC4 (128-bit):** Encrypts the PDF file with RC4 encryption. **WARNING:** This option is not recommended for extremely confidential data. Use the AES option instead.
 - **AES:** Encrypts the PDF file with the Advanced Encryption Standard. **NOTE:** If you choose this encryption method, Adobe Acrobat 7 or higher must be used to open the PDF file, as earlier versions of Acrobat do not support AES encryption.
- Back:** Displays the File Info screen.
- Finish:** Saves the export settings and closes the Export Wizard.
- Cancel:** Cancels any changes to the export's settings and closes the Export Wizard.

PRF Info Screen

Use this dialog to specify settings for a portable report. To access this dialog, select the Portable Report (PRF) as the file type when you set the folder and file name in the Output File dialog and then select Next.

The **Portable Report Settings** dialog includes the following options:

- Description:** Displays a description of Portable Report File: file name, file location, and behavior when output files exist (overwrite, add, or skip).
- PRF options:**
- **Include Tree View Index:** Includes the Tree view index in the portable report. Typically, the tree view index will contain the values from fields at each sort level in the report. When a portable report is viewed in Monarch Report Explorer, the tree view index is displayed in the same manner that Windows Explorer displays a directory tree. Each report represents a root of the tree, with branches for each field selected as part of the tree view index.
 - **Include Model:** Includes the model file in the portable report.
 - **Encrypt PRF:** Password-protects the portable report.

- ❑ **Change PRF Password:** Specifies a password for the PRF via the Change PRF Password dialog.
- ❑ **Compression options:** Select one of the following compression options:
 - **None:** Specifies no compression of the portable report contents.
 - **Medium (Fast):** Compresses and decompresses slightly faster than the Maximum compression option, and yields a compression ratio of approximately 5:1. This compression ratio may vary depending upon the density of information in your reports.
 - **Maximum:** Compresses and decompresses slightly slower than the Medium compression option but typically yields a compression ratio of 10:1. This compression ratio may vary depending upon the density of information in your reports.

NOTE If you plan to distribute the portable report you should compress it by selecting the Medium or the Maximum option.

- ❑ **Back:** Displays the [Output File](#) screen.
- ❑ **Finish:** Saves the current export settings and closes the Export Wizard.
- ❑ **Cancel:** Cancels the export operation and closes the Export Wizard.

Tableau Server screen

This screen displays when, during the configuration of a project export, you select **TDE** from the **Extension** drop-down list on the **Output File** screen of the **Export Wizard**, then click the **Next** button. Use this screen to specify settings of the **Tableau Server** for publishing Tableau export to the **Tableau Server**.

The **Tableau Server** screen includes the following options:

- ❑ **Publish to Tableau Server:** Select this option to display Tableau Server settings.
- ❑ **Data Source Name:** The name of the Data Source on the Tableau Server.
- ❑ **Path to TABCMD:** The route to the executable file of the TableauServerTabcmd application.
- ❑ **Server Address:** The address of the Tableau Server.
- ❑ **Username:** The login name to the Tableau Server.
- ❑ **Use Password File:** Allows to use the .txt file with the password.
- ❑ **Password:** The password to the Tableau Server (if the option Use Password File enabled, it allows to specify the path to the Password File).
- ❑ **Proxy Address:** The address of the proxy server (**NOTE:** you must use HOST:PORT format).
- ❑ **Publish Type:** Allows to specify an action for the Data Source:
 - **overwrite data source:** overwrites data in Data Source by data from the export table (**NOTE:** date modified is changed);
 - **replace data:** replaces data in Data Source with data from the export table (**NOTE:** date modified is not changed);
 - **append data:** adds data from the export table to the end of the Data Source table.
- ❑ **Back:** Displays the [Output File](#) screen.
- ❑ **Finish:** Saves the current export settings and closes the Export Wizard.
- ❑ **Cancel:** Cancels the export operation and closes the Export Wizard.

OleDb Info Screen

The **OleDb Info** screen is part of the Export Wizard, and displays when you select OleDb provider from Table View. Use this screen to specify the OLE DB or ODBC data source you want to connect to.

The **OleDb Info** screen includes the following settings:

- Displays a description of what the current export will do.
- Connection String:
 - Enter the connection string manually, or
 - click the Browse for ... button and select Data Link Wizard to open Data Link Wizard window. For more information see [Connection Type Wizard](#), or
 - click the Browse for ... button and select Connection Definitions to open Connection Definitions window and select from existing connection definitions. For more information, see [Connection Definitions](#)

After you made your choice the **Connection string** field displays the connection string for the data you want to connect to.

- Select Include Password check box if your connection requires password. For security reasons, the password is displayed here.

NOTE Passwords are saved in an encrypted format using 1024-bit encryption strength.

- **Test Connection:** Tests whether or not the data link was successful (i.e., to test whether you can connect to the specified OLE DB or ODBC provider). If the connection is successful, a Test Connection Succeeded message is displayed.
- Back:** Displays the [Table View](#) or [Summary View](#) screen.
- Next:** Displays the [Table Info](#) screen.
- Cancel:** Cancels the export operation and closes the Export Wizard.

Table Info Screen

The **Table Info** screen is part of the Export Wizard. When configuring an export, use this screen to specify the table to which you want to export data, and to select a When output tables exist option, which tells Automator how to handle an already existing table of the same name.

The **Table Info** screen displays when you click the Next button on either the Output File/Folder screen or the OleDb Info screen when exporting to a database table (xls or mdb).

The **Table Info** screen includes the following settings:

- Description:** Displays information about the export, such as whether filters are applied to the table or summary data, the filename and path of the output file, etc.
- Table Name:** Displays the current name of the table. You can assign a new table name by entering it here.
- Existing Tables:** Displays the names of the tables in that file when exporting to a file that already exists.
- When output tables exist options:
 - **Overwrite existing table:** Overwrites the data in the existing table with the data in the export table.
 - **Append data to existing table:** Adds the export table data to the end of the selected existing table.
 - **Update existing rows** (OLE DB direct exports of table data only): Updates existing rows in a table.
 - **Update existing rows and append new** (OLE DB direct exports of table data only): Updates existing rows in a table and append new rows to it.

- **Skip** (no output): Skips the export operation if a table of the same name already exists. This is useful for automated operations, where you may not want to overwrite a table that already exists.

NOTE Some of these options apply only to OLE DB direct exports. In addition, if you are performing a file system output, some of these options are only accessible if you selected the **Add data to file** option on the **File Info** screen.

- **When update match not found:** Select the Export to delimited text file check box to export the unmatched record(s) to a delimited text file when updating existing rows in a table, if a matching row is not found in the destination table. The path of this file will be written to the job log under an Exceptions heading (below the Outputs section).
- **Back:** Displays the File Info screen or the OleDb Info screen.
- **Next:** Displays the [Column Info](#) screen, if you selected an existing table to export to under the Existing Tables heading (see above).
- **Finish:** Saves the current export settings and closes the Export Wizard, if you are exporting to a new table.
- **Cancel:** Cancels the export operation and closes the Export Wizard.

Column Info Screen

The **Column Info** screen displays when you click the **Next** button on the Export Wizard's [Table Info](#) screen.

NOTE If you are updating existing rows in a table, you must map at least one key field and one data field. To designate a key field, right-click on a mapping and select **Enable** as key field. A key icon will appear to the left of the field name.

The **Column Info** screen includes the following settings:

- Export all:** Exports all the columns in the export table.

NOTE When appending data to an existing table, appends will only occur for columns that exist in both the export table and the destination table.

- Allow unrestricted mapping so that any Export column may be mapped to any Destination column:** Relaxes Automator's type checking and relies on the OLE DB Provider to perform type conversions. For example, if you need to export a numeric value that is so large that it cannot be stored in Monarch as a number, you can export it as a character field. The character field can then be mapped to a numeric field in DB2, for example, and the OLE DB provider will perform the type conversion, if that functionality is available.
- Export Column:** Lists the columns in the export table that have been mapped to columns in the destination table.
- Destination Column:** Lists the columns in the destination table that have been mapped to columns in the export table.
- Auto Map:** Instructs Automator to automatically map columns in the export table to columns in the destination table that have the same names. **NOTE:** Columns that don't occur in both tables will not be mapped.
- Add:** Displays the Select Columns to Map dialog, with which you can map an export column to a destination column.

- ❑ **Edit:** Edits the selected mapping via the Select Columns to Map dialog.
- ❑ **Remove:** Removes the selected mapping from the list.
- ❑ **Remove All:** Clears all mappings from the list.
- ❑ **Cancel:** Cancels any changes you have made and closes the Export wizard.
- ❑ **Back:** Displays the [Table Info](#) screen of the Export Wizard.
- ❑ **Next:** Displays the [Excel Advanced Features](#) screen when exporting to Microsoft Excel.
- ❑ **Finish:** Saves the current export settings and closes the Export Wizard.

Excel Advanced Features Screen

Use this dialog to use Excel Advanced Features when exporting to Microsoft Excel.

The **Excel Advanced Features** dialog box includes the following settings:

- ❑ **Description:** Displays information about the export, as entered in previous screens. For instance, if you are exporting a Summary, the description displays the summary being exported, the output file or folder to export to, and table that will be used.
- ❑ **Document Properties:** Displays the Document Properties dialog box. Here, you can set the property values of the Excel file, such as its Category, Creator, Description, Keywords, Subject, and Title.
- ❑ **Digital Signature:** Allows you to embed the currently selected digital signature to the exported Excel file (XLSX/XLSM). This option is available for Table and Summary exports. By adding a digital signature, you establish the authenticity, integrity, and non repudiation of the export output.
- ❑ **Certificate...:** Displays a Select Certificate dialog. Options will allow you to digitally sign with a valid X.509 Certificate (an existing Certificate selected from the Current User's Certificate Store or from a Certificate file).
- ❑ **Add AutoFilter drop down lists to column labels in the Table:** Enables the Excel AutoFilter feature, ready and active when you open the exported Excel file. This option is available for Table exports.
- ❑ **Include Report view text in XLSX file:** Allows you to include source data with the exported Excel file (XLSX/XLSM). This option is available for Table exports. Readers of the exported file can work with the Monarch Context add-in to reference the source data, or the even view the source report.
- ❑ **Include Pivot Table sheet:** This option is available for Table exports to Excel 2007 (xlsx) files.
 - **Fields:** Use this tab to select the desired pivot field filters, columns, rows and values.
 - **Options:** Use this tab to specify the Pivot Table name, action (if the Pivot Table exists), style, and grand total parameters.
- ❑ **Back:** Displays the [Column Info](#) screen.
- ❑ **Finish:** Saves the current export settings and closes the Export Wizard.
- ❑ **Cancel:** Cancels the export operation and closes the Export Wizard.

Specifying Default Export File Types

When configuring an export for a project, you have the option to specify default export file types. If you don't specify default export file types, the defaults stored in the Registry will be used.

NOTE The export file types you specify are stored in the project file. They are applied only to exports that utilize the current project file.

To specify the default version for each supported export file type when exporting table or summary data for the current project, click the **Types...** button on the Project Editor's Exports tab.

NOTE This dialog does not include options for XML-SS, HTML, Fixed Text (TXT), and Delimited Text (typically CSV), as only a single version of each file type is supported by MDP.

The **Export File Types** dialog box includes the following settings:

- Override registry settings:** Overrides the default export file types currently specified in the Registry.
- DBF Files:** Displays the file format and version to be used when exporting dBASE DBF files. The drop-down list includes all versions of the dBASE DBF file format supported by the Jet 4.0 database engine. **NOTE:** The drop-down list also includes a dBASE III (Monarch V3) entry. This option is available for users who have upgraded from Monarch version 3 and who have a need to create dBASE III export files with the exact same structure as those created by Monarch version 3. However, this option should not be required in most cases, as the Jet dBASE III format is only slightly different in structure from the Monarch V3 dBASE III format.
- MDB Files:** Displays the file format and version to be used when exporting MDB (MS-Access) files. The drop-down list includes all versions of the MS-Access MDB file format supported by the Jet 4.0 database engine.
- OK:** Saves the current settings to the Registry (if the Override Registry Settings option is checked) and closes the dialog.
- Cancel:** Cancels any changes you have made to the settings and closes the dialog.

Specifying Prerequisite Settings for a Project File

When configuring a process, you have the option to specify prerequisites for the associated project file(s), which must be met if the process is to run.

To specify process prerequisites

1. On the Navigation Tree, click Automator > Standard Processes and select Project Editor.
2. On the **Prerequisites** tab, specify the type of prerequisite you want to add, either **File** or **Command line**, by clicking the corresponding button. The Add Prerequisite dialog box is displayed.
3. In the **Add Prerequisite** dialog box do one of the following:
 - To specify a file prerequisite, specify a file name in the corresponding field or click the Browse button to select it via the dialog that displays, and then click the OK button. The path and filename of the file prerequisite appear under the Prerequisite heading.
 - To specify a command line prerequisite, specify a command line or click the Browse button to select it via the dialog that displays and enter an integer success code in the corresponding fields, and then click the OK button. The command line and success code appear under the Prerequisite and Success Code headings on the Prerequisites tab.
4. Perform Steps 2 and 3 again to add any additional prerequisites, then click the **OK** button to save the prerequisite(s).
5. To test added prerequisites do one of the following:
 - If machine location is specified in the prerequisite path, click the Test button adjacent to the path field.
 - If machine location is not specified in the prerequisite path, then select it from the Location for Testing drop-down list and click the Test button adjacent to the prerequisite path field.
6. To save or download created project, click the corresponding button or continue specifying other project settings if necessary.

NOTE When saving a project file provide a comment in the displayed window and click **Commit**. The Server Library [Project page](#) is displayed, where you can edit project file name, revision and access rights if necessary.

When the scheduled run time arrives, Automator will check the specified prerequisites to see that they have been met, and if they have been, the process will run. If they haven't, the process won't run.

Specifying Script Settings for a Project File

When you specify script settings for a project, you add a project-specific script to it.

NOTE This step is optional. If you have no need to utilize scripts in your project, you may skip this step.

Automator users can only edit these scripts if they are granted editing privileges by the Admin.

To specify script settings for a project

1. On the Navigation Tree, click **Automator > Standard Processes**, select **Project Editor** and navigate to the Script Editor tab.
2. Step through the Script Editor's tabs to specify the desired script settings:
 - [PreExport tab](#): Specifies project-specific, pre-export declarations, e.g., to apply a pre-export script to the input file.
 - [PostExport tab](#): Specifies project-specific, post-export declarations, e.g., to add steps to further process the export file(s).
 - [Global Declarations tab](#): Specifies Dim statements or other statements that must be global to the entire script module.
 - [References and Imports tab](#): Specifies additional .NET assembly references and namespaces required by the PostExport, PreExport, and Global Declarations tabs.
 - [Passwords tab](#): Specifies passwords for use in password macros.
3. In the **Locations for testing** field, select location from the drop-down list.
4. To save or download created project, click the corresponding button or continue specifying other project settings if necessary.

NOTE When saving a project file provide a comment in the displayed window and click **Commit**. The Server Library [Project page](#) is displayed, where you can edit project file name, revision and access rights if necessary.

Once you have finished configuring the project file, you may [assign it to a process](#).

Specifying Login Credentials

The Login Credentials tab is part of the Project Editor. Use this tab to specify the HTTP, FTP/SFTP, and Amazon S3 login credentials to be written into the project file when using input files from an FTP/SFTP site, a SharePoint repository, or Amazon s3 storage.

NOTE Input files located on FTP/SFTP sites, SharePoint repositories, or Amazon S3 storage cannot be used if monitoring is enabled. In addition, there can only be one set of credentials per project, so you should not specify input files that reside on different locations which require different login credentials.

To specify login credentials

1. On the Navigation Tree, click **Automator** > **Standard Processes** and select **Project Editor**.
2. On the **Login Credentials** tab, do one of the following:
 - To specify HTTP login credentials, specify a name in the Login field and password by clicking the Password button.

NOTE To use the current windows login select the corresponding check box.

- To specify FTP/SFTP login credentials, specify a name in the Login field and password by clicking the Password button.
 - To specify Amazon S3 login credentials, specify an ID in the Access key ID field and an access key by clicking the Secret access key button.
3. To save or download created project, click the appropriate button or continue specifying other project settings if necessary.

NOTE When saving a project file provide a comment in the displayed window and click **Commit**. The Server Library [Project page](#) is displayed, where you can edit project file name, revision and access rights if necessary.

VISUAL PROCESSES

Automator has a Visual Process Designer tool that allows you to create the process and define its work flow in a fast and convenient way. The Visual Process Designer visualizes mechanism of process components assembling by providing you with the library of various graphic elements and connectors and a workspace within which you can arrange them. Besides, the Process Designer can intuitively arrange the elements on the workspace.

Information on starting and finishing processes when executing processes are stored in process logs.

Processes

All the existing Visual Processes are displayed on the Processes page. To view the **Processes** page, on the Navigation Tree, click **Automator > Visual Processes** and select **Processes**.

The **Processes** page shows the following:

- Process name
- Logs column linked to the Process Logs page
- Process description
- Process current status displayed in the Usage column:
 - Monitored enabled
 - Monitored disabled
 - Scheduled enabled
 - Scheduled disabled
 - Ad hoc (i.e., no monitoring or schedule defined)
 - Report Warehouse Input enabled
 - Report Warehouse Input disabled
- Date and time the process was created
- Name of the author who created the process
- Date and time the process was updated
- Date and time the process was last run
- Most recent status of the process

NOTE When deleting a visual process that uses a model, workspace, or change list in the Server Library, a dialog box appears and gives the option to **Delete dependencies**. Clicking the **Delete dependencies** button removes the associated model, workspace, or change list from the Server Library. Clicking **Cancel** deletes the process only and not the associated model, workspace, or change list. The **Delete dependencies** option is not available when the model, workspace, or change list is used by more than one process.

From the Processes page, you may also

- [Import Processes](#)
- [Export Processes](#)
- [Migrate Standard Processes](#)

NOTE Using the corresponding fields in the column headings, you can filter the processes by

- Name
- Description
- Usage

- Created
- Author
- Updated
- Updated by
- Last run
- Last status

The following filter settings are available:

- ~ - contains
- == - equal
- < - less
- > - greater
- X - Reset search value

Adding a Process

To add a Process

1. On the Navigation Tree, click **Automator** and select **Visual Processes**.
2. On the **Processes** page, click the Add icon . The [Visual Process Designer interface](#) will be displayed.
3. Choose Blank Process to work on a Visual Process from scratch. To use a template, you may choose from the following:
 - Data Prep
 - Distribute export result
 - Groups
 - Model verify
 - Parallel export
 - Script
 - Sequential export

Editing a Process

Once you have configured and saved a Visual Process, you can edit it at any time.

To edit a process

1. On the Navigation Tree, click **Automator** and select **Visual Processes**.
2. Click the name of the process you want to change or clicking anywhere on its row, and then click . You can then modify the process from the Visual Process Designer interface.

NOTE

When loading a Visual Process containing a workspace element, workspace validation is initiated and is terminated automatically after a certain number of seconds. The duration of workspace validation can be modified in Automator's

[General Settings](#). You may terminate the workspace validation manually by clicking Cancel in the loading screen.

Deleting a Process

You can delete any process on the Processes page.

To delete a process

1. On the Navigation Tree, click **Automator** and select **Visual Processes**.
2. Click at the end of the row with the process you want to delete. A dialog box appears.
3. Click **Delete** to remove the process from the list.

The process is removed from the [Processes](#) page, and its associated logs are removed from the [Logs](#) page.

NOTE When deleting a visual process that uses a model, workspace, or change list in the Server Library, a dialog box appears and gives the option to Delete dependencies. Clicking the Delete dependencies button removes the associated model, workspace, or change list from the Server Library. Clicking Cancel deletes the process only and not the associated model, workspace, or change list. The Delete dependencies option is not available when the model, workspace, or change list is used by more than one process.

Enabling/Disabling Monitoring of Visual Processes

You can enable or disable the monitoring of multiple Visual Processes simultaneously.

To enable/disable process monitoring

1. On the Navigation Tree, click **Automator**, and select **Visual Processes**.
2. Select the Visual Processes you want to enable/disable the monitoring for by clicking their corresponding checkboxes.
3. Click the drop-down arrow , and choose **Disable Monitoring** or **Enable Monitoring**.
4. In the **Monitoring** dialog box, click **Ok** to proceed.

NOTE When enabling/disabling monitoring or changing the path of a monitored folder, the changes take effect after approximately 1 minute.

Enabling/Disabling Scheduled Visual Processes

You can enable or disable one or more scheduled Visual Processes.

To enable/disable scheduled Visual Processes

1. On the Navigation Tree, click Automator, and select Visual Processes.

2. Select the Visual Processes whose schedule you want to enable/disable by clicking their corresponding checkboxes.
3. Click the Additional schedule actions icon , and choose Disable Schedules or Enable Schedules.

NOTE When enabling/disabling schedules, the schedule status (i.e., active/suspended) of the selected processes is automatically saved.

Visual Process Designer

The Visual Process Designer tool allows you to create processes in easier and more convenient way than to apply the standard approach. You have full set of tools for process creation that is used in standard processes but you have greater capabilities to apply these tools and therefore set up more sophisticated scenarios.

Data acquisition, conversion and distribution is visualized in the Process Designer through use of graphic elements and connectors arranged by threads.

By using library of graphic elements and a workspace, you can quickly assemble different process components to set the required process. The created process can be represented as a graph in the Process Designer window.

Visual Process Designer Interface

The Visual Process Designer interface consists of upper line, Process Designer Library, Workspace panel, and Properties panel.

The upper line of the Process Designer contains name of the process (on the left) and the Designer Toolbar. You are allowed to change name of the process if needed.

The table below specifies buttons of the Visual Process Designer toolbar.

BUTTON NAME	DESCRIPTION
New	Click to open new workspace for creation new process. The workspace with the previous process will not be saved automatically.
Save	Click to save the specified work flow as a process.
Save as ...	You can choose the following options: <ul style="list-style-type: none"> <input type="checkbox"/> Save new process: Save the specified work flow as a new process. <input type="checkbox"/> Save as template: Save the specified work flow as a template. Elements settings are not saved. <input type="checkbox"/> Save as template (retain data): Save the specified work flow as a template. Elements settings are saved.
Insert Template	Click to open the list of existing templates. You can select the required template and add it to the workspace.
Insert Process as Group	Click to open the list of existing processes. You can select the required process and add it to the workspace as a group element.
Triggers	Click to specify when the process will run. You can choose the following types of process triggers: <ul style="list-style-type: none"> <input type="checkbox"/> Schedules <input type="checkbox"/> Inputs Monitoring

BUTTON NAME	DESCRIPTION
	<p data-bbox="488 247 760 317"> <input type="checkbox"/> Manifests Monitoring <input type="checkbox"/> Report Warehouse </p> <p data-bbox="488 333 938 361">Only one of them can be chosen at a time.</p> <p data-bbox="488 422 751 449">SCHEDULES TRIGGER</p> <p data-bbox="488 459 1252 537">A schedule is used to specify when, and at what interval the process will run. You can create a number of schedules but their launch time must differ.</p> <p data-bbox="488 583 1263 747">NOTE: If the Config service is not available and you are using Altair licensing, Automator processes will be switched to "Limp Mode." In this mode, you can run only one process (i.e., one Standard Process and one Visual Process) every hour. Moreover, you cannot run a Visual Process that involves other Standard or Visual Processes. Limp Mode is deactivated once the Config service becomes available.</p> <p data-bbox="488 779 907 806">To add a new schedule for a process</p> <ol data-bbox="488 831 1252 1188" style="list-style-type: none"> 1. Click the Triggers icon. 2. On the Process triggers dialog, select the Schedule radio button. 3. Click the Add schedule button. 4. Select the task execution frequency from the following options: One Time, Hourly, Daily, Weekly, Monthly. 5. Set the time you want the process to run. 6. Set the start date of the process. 7. Select the appropriate time zone from the drop-down list. 8. Click the Ok button to save the schedule. <p data-bbox="488 1220 724 1247">To delete a schedule</p> <ol data-bbox="488 1272 1159 1386" style="list-style-type: none"> 1. Click the Triggers icon. 2. Click the Delete icon near the schedule you want to delete. 3. Click the Ok button to save process schedules. <p data-bbox="488 1402 1187 1430">Select the Suspend check box to temporarily hold up the trigger.</p> <p data-bbox="488 1497 854 1524">INPUTS MONITORING TRIGGER</p> <p data-bbox="488 1539 1276 1652">When you specify inputs monitoring settings for a process, you specify files that you want Automator to monitor a particular location for. When the file arrives in the specified location, Automator will trigger running of the associated process.</p> <p data-bbox="488 1667 1252 1745">NOTE: Changes in monitoring settings, including enabling and disabling monitoring or changing paths for monitored folders, take effect after approximately 1 minute.</p> <p data-bbox="488 1776 894 1803">To specify an inputs monitoring file</p> <ol data-bbox="488 1829 1260 1896" style="list-style-type: none"> 1. Click the Triggers icon. 2. On the Process triggers dialog, select the Inputs Monitoring radio

BUTTON NAME	DESCRIPTION
	<p>button.</p> <ol style="list-style-type: none"> To add a file from the process, expand the Process inputs sections and press the Add icon near the input path, which you want to add to the Monitoring Paths list. To add a file not from the process, expand the Browse section, set the file path, and click the Add button. <p>NOTE: If you're unsure of the file's exact name or if you want Automator to monitor for any file of a particular file type, you can use a wildcard character, either an asterisk (*) or a question mark (?), so that the arrival of any file of the specified type will trigger the running of the process. An asterisk may replace multiple characters while a question mark may replace any single character.) For example, adding *.prn to the end of the folder path instructs Automator to trigger the running of the associated process when any report file with a .prn extension arrives in the specified location.</p> <ol style="list-style-type: none"> Expand Options to specify monitoring behavior: <ul style="list-style-type: none"> Disable monitoring if trigger file is not removed – The input mask (e.g., myfile.txt or *xyz.pdf) from the monitored folder can sometimes cause a process job to fail with an error and remain in its original folder. By selecting Disable monitoring if trigger file is not removed, Automator stops monitoring the corresponding folder(s) for the process and its input mask(s) only. When this option is not selected, the monitoring is left enabled; when new files appear in the monitoring folder, the process running is triggered again. You would, for example, enable this option if a problem requires intervention to determine why the inputs caused the process to fail and prevent the process from running until the issue is addressed. All files must exist – Monitoring based on multiple files starts as soon as one of the files is found. By selecting the <i>All files must exist</i> option, the monitoring process will not start unless all the associated files are found. Click the Ok button to save the process monitoring. <p>To delete an inputs monitoring file</p> <ol style="list-style-type: none"> Click the Triggers icon. Select the Inputs Monitoring radio button. Click Delete icon near the file path you want to delete. Click the Ok button to save process monitoring. <p>MANIFESTS MONITORING TRIGGER</p> <p>When you specify manifests monitoring settings for a process, you specify the manifest XML files that you want Automator to monitor a particular location for. When the file arrives in the specified location, Automator will trigger running of the associated process.</p> <p>To specify a manifests monitoring file</p> <ol style="list-style-type: none"> Click the Triggers icon. On the Process triggers dialog, select the Manifests Monitoring

BUTTON NAME	DESCRIPTION
	<p>radio button.</p> <p>3. To add a file, expand the Browse section, set the file path, and click the Add button.</p> <p>NOTE: If you're unsure of the file's exact name or if you want Automator to monitor for any file of a particular file type, you can use a wildcard character, either an asterisk (*) or a question mark (?), so that the arrival of any XML file will trigger the running of the process. An asterisk may replace multiple characters while a question mark may replace any single character.)</p> <p>4. Click the Ok button to save the process monitoring.</p> <p>To delete a manifests monitoring file</p> <ol style="list-style-type: none"> 1. Click the Triggers icon. 2. Select the Manifests Monitoring radio button. 3. Click Delete icon near the file path you want to delete. 4. Click the Ok button to save process monitoring. <p>REPORT WAREHOUSE TRIGGER</p> <p>This trigger is available only if in the Monarch Server License the Enterprise Server component is enabled.</p> <p>It enables you to specify certain document types so, when the system receives a report with the corresponding document type, the trigger runs a process.</p> <p>To assign Report Warehouse document types</p> <ol style="list-style-type: none"> 1. Click the Triggers icon. 2. Select Report Warehouse. 3. Add document types by clicking the plus icons or by drag and dropping them into the Assigned Document Types field. 4. Click Ok. <p>You can also add document types from the Report Warehouse Input of the current process by clicking the Assign from process button.</p> <p>Select the Disable Triggers check box to temporarily turn off all triggers.</p>
Scripts	<p>Click to apply scripts to the current process.</p> <p>In the Process Declarations tab, the following settings are available.</p> <ul style="list-style-type: none"> <input type="checkbox"/> Script field: Displays the existing process declarations. <input type="checkbox"/> Insert Script: Allows you to quickly generate VB.NET script for frequently used tasks. For complete options, refer to Inserting a Script section. <input type="checkbox"/> User Scripting Help: Opens the User Scripting API Help, which is a collection of script fragments that can be copied and pasted as useful starting points in the Script Editor. <p>In the References and Imports tab, specify additional assembly references and namespaces that are required by PostExport, PreExport,</p>

BUTTON NAME	DESCRIPTION
	<p>and Global Declarations sections. The following settings are available:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Namespace: Lists the namespaces you have specified. You can add a namespace by clicking the Add button. <input type="checkbox"/> Name: Lists the names of any additional assembly references you have specified. To add a name, click the Add button. <input type="checkbox"/> Assembly: Lists the paths of any assemblies you have selected.
Generate Manifest File	<p>Click to create a manifest XML or JSON file that is based on the settings of the visual process. The created XML/JSON file can be used immediately to initiate process runs.</p> <p>In the Manifest window, select the preferred manifest file format:</p> <ul style="list-style-type: none"> <input type="checkbox"/> XML – creates a manifest XML file <input type="checkbox"/> JSON – creates a manifest JSON file <p>NOTE: This icon is available only after the visual process is saved.</p> <p>NOTE: Before starting a process with a manifest containing some passwords, make sure to specify all required passwords and uncomment the “TODO:Specify Password” row in the manifest. If the process has no passwords, the manifest file generated automatically could be retained as it is.</p>
Alerts	<p>Click to view, add, edit, or remove alerts for the current process.</p> <p>In the Process Alerts window, click the Add icon  or the Edit icon  to create or edit an alert, respectively. The following settings are available:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Code: Specify the alert type. <input type="checkbox"/> Description: Specify the alert description. <input type="checkbox"/> Ignore Global Alert: Select this check box to ignore global alerts. <p>NOTE: This option is available only when the Allow ignore Global alerts setting is enabled. See Automator Settings for details.</p> <ul style="list-style-type: none"> <input type="checkbox"/> Immediate alert: Select the Raise the alert immediately when detected within running process check box if the alert being created should be immediate. <p>NOTE: The Rise Alert Immediate checkbox is available only for the SystemAlertNoInputToExporter alert and the SystemAlertProcessElementFailed alert.</p> <ul style="list-style-type: none"> <input type="checkbox"/> Actions: Choose which action to add to the alert. The Log, Email, and RSS options are available. See Adding an Alert.
Save and Run	Click to save the project and launch its execution.
Stop	Click to cancel the execution of the project.
Auto Arrange	Click to let the Process Designer arrange graphic elements intuitively on your workspace.
Zoom In (+)	Click to increase the size of some workspace area so that you can focus on areas of particular interest.
Zoom Out (-)	Click to decrease the size of some workspace area so that you can return to a larger scope.

BUTTON NAME	DESCRIPTION
Clear	Click to clear Visual Process Designer workspace.
Log Mode	<p>Click to open the Log Mode view.</p> <p>For a new process that has not been run yet, there are no logs available. Otherwise, the Log Mode view shows the results of the last run of the current processes.</p> <p>The Log Mode allows visual elements to be edited, along with their respective settings. After changes are made, click the Save and Run icon to actualize your changes by running a process with updated settings. Click Stop to cancel the execution of the process.</p> <p>Process Events displays general information about the recently run process, including the start and end times.</p> <p>Process Items allows you to filter the process information by selecting and clearing the following check boxes:</p> <ul style="list-style-type: none"> <input type="checkbox"/> All — displays all file sources. <input type="checkbox"/> Warning — displays the file sources with warnings. <input type="checkbox"/> Error — displays the files sources with errors. <p>Click the n Events link below each file source to expand the relevant information, including the start and end times.</p> <p>The items may be sorted according to duration (ascending and descending), start time (ascending and descending), and type.</p> <p>Click the Export Log button  to export the log to an XML file.</p> <p>Click the Designer Mode icon to leave the Log Mode view.</p>
Process Global Settings	<p>Click to modify the workspace validation settings of the Visual Process in the Process Global Settings window.</p> <p>Select the Disable Validation checkbox to turn off the workspace validation for the loaded Visual Process.</p> <p>NOTE: This setting has higher priority than the workspace validation setting in Automator General Settings.</p>
Rights	<p>Click to assign process access rights to other users, roles, or groups.</p> <p>NOTE: See Application Settings for more details.</p>
Help	Check for information about basic and advanced Visual Processes.
Close	Click to close Visual Process Designer.

Major part of toolbar controls can be called using the context menu. The Process Designer context menu is called on the right click on the workspace area. You can use the following commands from the Process Designer context menu:

- Insert Template
- Insert Process as Group
- Auto Arrange
- Zoom In
- Zoom Out

- Show/Hide Grid
- Enable Auto Connect
- Show advanced connectors
- Clear

The context menu is also available for all graphic elements. You can use the following commands from the element context menu:

- Duplicate** — to duplicate the selected element.
- Group** — to combine several selected elements into a single element group (available for multiple selected elements only).
- Ungroup** — to revert elements grouping (available for selected group elements only).
- Drill in** — to open the group element (available for group elements only).
- Rename** — to rename the selected element.
- Delete** — to delete the selected element.
- Properties** — to open properties of the selected element.

On the left panel, there is a Process Designer Library with graphic elements to denote different types of inputs, models, exporting jobs, prerequisites, etc. All items from this library can be used to assemble the required process.

To the left of the Process Designer Library, the Workspace panel is adjusted to where you can drag and drop graphic elements from the Library panel.

Single or double click on a particular graphic element opens panel with its properties. When displayed, the Property panel is located on the right part of the Process Designer window. Mandatory buttons of the Properties panel are as follows:

- Ok** — to save the changes made on the Properties panel and close the panel.
- Apply** — to save the changes made on the Properties panel and leave the panel to be expanded.
- Cancel** — to cancel the changes made on the Properties panel.

NOTE When you do not click **Ok** or **Apply** in the Property panel and you click on another graphic element in the Workspace panel, a dialog box appears to warn you about losing unsaved changes. In the dialog box, you can choose to save or not save the changes you have made in the Property panel. Clicking **Cancel** will close the dialog box, and the Property panel will remain open.

The Process Designer Library contains the following groups of the graphic elements:

- [Input](#)
- [Model](#)
- [DataPrep](#)
- [Export](#)
- [Prerequisites](#)
- [Misc](#)
- [Output](#)

Some library elements define the process work flow, and some not. For example, library elements from the **Input** and **Model** groups represent data, but not work flow.

Process work flow is defined by graphic elements in the following library groups:

- Exports
- Prerequisites
- Misc (with the exception of the Credential element)
- Output

Elements from the **Input** and **Model** groups do not define process work flow. Belonging to work flow is a benchmark why two types of connectors are used, namely:

- Solid line connectors** — for connecting work flow items
- Dashed line connectors** — for connecting non-work-flow items.

One more work flow (not data) indicator for a graphic element is availability of input and output pins of gray color on the element's icon ().

Connectors differ by color:

- Gray connector** — implementation of work flow stage is successful
- Red connector** — implementation of work flow stage leads to failure

Quick adding the connected elements

Quick connectors are available upon mouse over an element — two plus signs (+) appear on top of it.

- Left plus connector** — allows to promptly select and add another input element connected to the current one.
- Right plus connector** — allows to select and add an output connected element.

NOTE Quick connector buttons for workspaces with selected settings of load plans and input files allows to select and add a connected input file element (unpinned input files only).

Group element

You can open the group element by right-clicking and choosing Drill In, or by double-clicking the element. After opening, the breadcrumbs line displays the group name.

If you want to exit the group, click the previous element in the breadcrumbs line.

You can add new elements to the group element by dragging and dropping.

You can connect the group element with the required element by dragging the connector icon from the group element to the required element. Also, you can connect the required element with an element of the group. After that, you can open the group element and proceed with connecting it to the necessary element in the group (the icon of that connection is selected and the name of the connecting element is displayed in the tool tip).

Instructions for process work flow building

- There are no mandatory graphical elements that must be used for work flow building.

- ❑ Starting work flow item is defined by absence of input connector. Process execution starts with the starting work flow item.
- ❑ Final work flow item is defined by absence of output connector. Process execution finishes on processing final work flow item.
- ❑ If there two or more starting items, execution starts synchronously and is divided in two parallel threads.
- ❑ You can fork work flow using connectors if you need to branch one thread of process execution into two or more parallel threads.
- ❑ Successful execution of work flow stage is defined by gray connector, unsuccessful execution of work flow stage is defined by red connector.
- ❑ By pop-up colorful arrows, the Process Designer gives prompts that some work flow items are available for connecting with the particular item and some work flow items cannot be connected with the given item.
- ❑ By pop-up single/double arrows, the Process Designer gives prompts that a particular work flow item can be connected with the single item only or with multiple items.
- ❑ For export operations, output work flow item can be an input item for subsequent work flow item.

NOTE

A Visual Process can be run in parallel with itself. This feature is useful in specific cases, e.g., those initiated by a manifest file with different inputs and outputs for each process run or when a macro output is used according to different input file names.

However, running the same Visual Process in parallel could lead to resource contention. This feature does not include any mechanism for locking inputs or outputs if they simultaneously appear in more than one instance.

Every time you run a process, process log item is created. Refer to the [Process Logs](#) section for details.

If you save some process as a template, it will be saved in the list of templates and could not be found in the list of processes. In this case, settings for all data flow items will not be saved.

Input Elements

Input elements in the Process Designer tool are data sources which are used for exporting.

The table below specifies functionality of the **Input** group elements.

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
File Input	Source text file in some file system	<ul style="list-style-type: none"> ❑ Name — name of the graphic element. ❑ Input Path — path that shows the source file location. <p>The Input Path parameter can be filled out manually or using Browse.</p> <p>Actions:</p> <ol style="list-style-type: none"> 1. Click the Browse button to open the Browse dialog. 2. Select the data source type in the Source Type drop-down list (currently, Local or Named Paths). 3. Depending on the selected data source type, specify the other parameters:

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		<ul style="list-style-type: none"> • For the Local type: Location of the input file and Path to this file using the suggested folder directory. • For the Named Paths type: Named Paths and file input path using the Named Path Content box (use search capabilities if needed). <p>NOTE: Input files in compressed archive format (e.g., zip and gzip) can be used and specified.</p> <p>4. Click the OK button.</p>
Web Input	Text file located in the web (FTP/SFTP, HTTP location, HTTPS, Amazon S3)	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Input Address — URL of the source file. <p>In the Input Path field, naming macros may be used. To do this, click the Insert Macro icon and select the naming macros from the drop-down list. (NOTE: The ability to use naming macros is only available for Visual Processes.)</p> <p>NOTE: If some web input file requires authentication, you should add the Credentials element and connect your Web Input element with it. In properties of the Credentials element you should specify the User Name and Password parameters.</p>
Database Input	Database file	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Data Source — data source to be determined. Specify this parameter manually or click the Browse For... button to select one of the following data source types: <ul style="list-style-type: none"> • Local or Web File • Data Link Wizard • Data Link File • Connection Definitions <p>Then select the required file using the displayed dialog.</p> <p>NOTE: In addition to Excel file types .xls and .xlsx, .xlsb and .xlsm file types may be used as database file input.</p> <p>If the input file is password protected, select the Include password check box and enter the required password in the corresponding field.</p> <p>To select a specific table within the input file to export data to, click the Table drop-down arrow and select the required table.</p> <p>To replace nulls in the table, select the Replace nulls with default values (empty strings, zeros, and default dates (1/1/1970)) check box.</p>

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		<p>To apply filtering to the table during the export, select the Apply Filter check box and click the Change button below to specify necessary filters.</p> <p>NOTE: Filters are not applicable when applied directly to a Workspace-based dataset. To implement a filter for a Workspace-derived export, apply a change list to the preceding Workspace object as part of the load plan selection.</p> <p>NOTE: When Automator exports the table data, the filter(s) you selected will be applied to it. Only the data that matches the filter's specifications will be exported.</p> <p>To include column names in the table with the exported data, select the First row contains column names check box.</p> <p>To choose an Excel engine, click on the Excel Engine drop-down list, and choose Legacy or Modern.</p> <p>NOTE: For the Modern Excel engine, password-protected Excel files can be used.</p> <p>Delimiter character — specifies the delimiter (i.e., comma, tab, semicolon, pipe, space, other) used in the file to separate values.</p> <p>If values may run beyond one line (i.e., the value includes a line break), select the Quoted values may contain embedded line breaks check box.</p> <p>Text qualifier — specifies what character is used (i.e., double quote, single quote, none) to enclose values. When Automator encounters this qualifier in a line, all of the text following this character and preceding its next instance is imported as one value, regardless of the delimiter used.</p> <p>NOTE: If the delimiter is a comma (,) and the text qualifier is a double quotation mark ("), e.g. "Boston, Massachusetts" is imported into one cell as "Boston, Massachusetts." By contrast, if no character or the single quotation mark (') is specified as the text qualifier, "Boston, Massachusetts" is imported into two adjacent cells as "Boston" and "Massachusetts."</p> <p>Character set — specifies the type of encoding to use when interpreting data.</p> <p>Lines to skip at start — the number indicated in this box instructs Automator to skip n lines before beginning data import.</p>

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
Report Warehouse Input	Reports from the Report Warehouse document type	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Document type — Report Warehouse data container the report input is taken from. Specify the document type of the reports to be considered as input. <p>You can search for documents using the following search criteria:</p> <ul style="list-style-type: none"> <input type="checkbox"/> By date: define a date range and then specify whether you want to find documents that were filed (registered) or documented (created) within this range. <input type="checkbox"/> Files from filing result that triggered the process: this option means the search will find only the report(s) that triggered the most recent process launch.

NOTE

If the source file has PDF format, the **PDF Password** field appears on the **Properties** panel. You need to enter password if the source file is protected with the password and click **Check**. The specified password for PDF file will be checked. If the password is wrong or you ignore password check, you cannot use the given input.

It is recommended to use wildcards in the **Input Path** field to enable using multiple input files or any input file of a particular file type. Use asterisk (*) to replace multiple characters and question mark (?) to replace some single character. For web input files, wildcards can be used only if it is an FTP/SFTP connection, not HTTP.

Model Elements

Model elements in the Process Designer tool are sets of instructions how input files are parsed by Automator.

A model file, created within Monarch, is a file with the *.dmod* or *.xmod* extension that may contain field, filter, summary and chart definitions as well as other settings.

The table below specifies functionality of the **Model** group elements.

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
File Model	Model file with the <i>.dmod</i> or <i>.xmod</i> extension	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Model Path — path that shows the File Model location. <p>The Model Path parameter can be filled out manually or using Browse.</p> <ol style="list-style-type: none"> 1. Click the Browse button to open the Browse dialog. 2. Select the data source type in the Source Type drop-down list (currently, Local or Named Paths). 3. Depending on the selected data source type, specify

		<p>the other parameters:</p> <ul style="list-style-type: none"> • For the Local type: Location of the model file and Path to this file using the suggested folder directory. • For the Named Paths type: Named Paths and the path to the model file using the Named Path Content box (use search capabilities if needed). <p>NOTE: Model files in compressed archive format (e.g., zip and gzip) can be used and specified.</p> <p>4. Click the OK button.</p>
Web Model	Model file located in the web	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Model Address — URL of the source model file. <p>NOTE: If some web model file requires authentication, you should add the Credentials element and connect your Web Model element with it. In properties of the Credentials element you should specify the User Name and Password parameters.</p>
Server Library Model	Model within a database	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Model Path — path to the Server Library Model source file. Specify this parameter manually or select the source file from the list displayed in the Server Library box. <p>NOTE: For external lookup from a password-protected Excel file, select the External Lookup checkbox and enter the password of the Excel file in the field provided.</p>

DataPrep Elements

DataPrep elements in the Process Designer tool are used for executing data preparation operations upon process inputs. These elements consist of Workspace and DataPrep Export.

NOTE A **Hub** element can be added between a **Workspace** element and a **DataPrep Export** element to process multiple input files individually or by group on a workspace.

The table below specifies functionality of the **DataPrep** group elements.

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
Workspace	Performs operations upon process inputs.	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Workspace Path — path to the required input file location. <p>Actions to select the workspace path:</p> <ol style="list-style-type: none"> 1. Click the Browse button to open the Browse dialog.

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		<ol style="list-style-type: none"> 2. Select among server library. 3. If you have slow data source connections, you can select the Delay data source connections validation check box. In this case, you can cancel the validation of Data source connections when loading workspace by clicking the Cancel button in the Properties panel. You can also cancel the validation of Data source connections when opening a process which contains a workspace. 4. Click OK. <ul style="list-style-type: none"> <input type="checkbox"/> Validate Workspace – Click this button to validate the workspace manually. The result of workspace validation shows the following status: <ul style="list-style-type: none"> • Validation successful • Validation failed • Validation is not completed <input type="checkbox"/> Load Plans — the list of available export load plans, including: <ul style="list-style-type: none"> • Provider type. This shows the type of data provider. • File Name. If your Workspace input is a file of a certain format, you can substitute it with the file from the process input. Do the following. For the PDF Report, Text Report and Delimited Text file formats, add a File input item to your visual process design form. For Microsoft Access and Microsoft Excel formats, add a Database input item. Connect this item with your Workspace element. Set up the Input path for your File input or Database input. Then in the File Name drop-down list for your load plan, select a File input or a Database input option. <p>NOTE: When the Database Input option is selected, the table name automatically changes to the table name from the database input. The table name may be changed manually.</p> <p>NOTE: DatabasInput is required for Excel formats and must include the extension even with wildcards.</p> <p>NOTE: Additional options are available for Excel Worksheet Design. They are detailed here.</p> <ul style="list-style-type: none"> • Source. If the workspace input is a database, then this field displays the database name. If the source type is SQL Server, ODBC, or OLE DB, there is the possibility to rewrite/edit the Database

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		<p>Data Source. To do so, click the Edit Table Info button, and then edit the Connection String, Table, etc. If the plugin is Google Analytics, the settings can be changed by clicking the Edit Table Info button. In the Google Analytics window, edit the Service Account Email, Profile ID, Start Date, End Date, etc. If the plugin is Oracle, edit the settings by clicking the Edit Table Info button. In the Oracle window, edit the connection type and settings (including host name, port, service name, user ID, password, etc.). Tables and views, as well as queries, can also be edited from this window.</p> <p>NOTE: Additional options for SQL Server, OLE DB, and Oracle source types are provided below.</p> <ul style="list-style-type: none"> • Change History. Additional operations to be executed with the input. Click the Add Change List button and select a change list from the Server Library to add to the workspace. You can also add new item to Server Library via the Add Change List dialog box. Upon selecting a change list, you will see a Change History list. You can add multiple change lists via the little Add Change List button on the right, or remove all change lists via the Reset Change History button. You can also disable certain changes to your workspace by clicking the respective Undo button (all the following change items will be disabled as well). Click the Redo button to enable the changes back. Non-applicable changes appear as disabled and accompany with a warning sign. You can remove such changes via the Delete button.
DataPrep Export	Exports data from data preparation workspace.	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Load Plan to be Exported — select a load plan for exporting. <p>The user can select several load plans for the export by ticking their corresponding checkboxes. To export all load plans, select the All Load Plans check box.</p> <ul style="list-style-type: none"> <input type="checkbox"/> Export Destination — path to required export file. <input type="checkbox"/> Export Type — the type of export file. Further settings depend on the type of file and are described as advanced options below. <p>NOTE: You may also use the naming macros to name the exported files and destination folder. To</p>

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		<p>do this, click the Insert Macro icon () and select the naming macros from the drop-down list.</p> <p>NOTE: The Input File Name macro is available to specify one or more input file names. If there are several inputs in a Load Plan (Join/Union), the names of the export files will contain a combination of these input files. When the Input files are processed by group, the Input File Name macro will only hold the first file name.</p>

Advanced options for Workspace element

Additional options for Excel Worksheet Design

1. When dealing with multiple inputs, the **Parse inputs for sheet names** option becomes available. Select the check box if you want to parse the inputs for specific sheets.
 - When selected, changes can be applied to the following:
 - ◆ All worksheets
 - ◆ First sheet
 - ◆ Last sheet
 - ◆ Selected Sheet by Name
 - When not selected, changes can be applied to the following:
 - ◆ Selected Sheet by Name
 - ◆ Selected Sheet by Number
2. When dealing with only one input, modifications can be applied to the following:
 - ◆ All worksheets
 - ◆ First sheet
 - ◆ Last sheet
 - ◆ Selected Sheet by Name

Additional options for SQL Server, OLE DB, and Oracle source types

1. In the column list in the SQL Server/OLE DB/Oracle dialog box, select the column to use.
2. Select from the following options:
 - **Aggregate:** Select the check box if the column data should be aggregated (i.e., grouped together) and then select the aggregation method to apply from the drop-down menu. The following aggregation methods are available:
 - ◆ **Sum:** Applies to numeric data; returns the total value of all values in the column
 - ◆ **Count:** Applies to numeric data and dates; returns the number of entries in the column
 - ◆ **Min:** Applies to numeric data and dates; returns the lowest value in the column
 - ◆ **Max:** Applies to numeric data and dates; returns the highest value in the column
 - ◆ **Group by:** Applies to numeric data, dates, and text; returns grouped values and is often used with functions such as SUM and COUNT.
 - **Apply Filter:** Select the check box if the column data should be filtered and then select the filter method to apply from the drop-down menu. Enter the filter value to use in the corresponding Filter Value box.

The following filter methods are available:

- ◆ **EqualTo:** Applies to numeric data, dates, and text; returns all values that contain the specified filter value
- ◆ **NotEqualTo:** Applies to numeric data, dates, and text; returns all values except those that contain the specified filter value
- ◆ **GreaterThan:** Applies to numeric data and dates; returns all values that are greater than the specified filter value
- ◆ **GreaterThanEqual:** Applies to numeric data and dates; returns all values that are greater than or equal to the specified value
- ◆ **LessThan:** Applies to numeric data and dates; returns all values that are less than the specified filter value
- ◆ **LessThanEqual:** Applies to numeric data and dates; returns all values that are less than or equal to the specified filter value
- ◆ **Contains:** Applies to text; returns all values that contain the specified filter value
- ◆ **StartsWith:** Applies to text; returns all values that start with the specified filter value
- ◆ **EndsWith:** Applies to text; returns all values that end with the specified filter value

Advanced options for DataPrep Export element

Additional options for Altair Knowledge Studio file types

1. In the **When output file exists** drop-down menu, select **Overwrite** to substitute files with the same file names.

Additional options for CSV file types

1. In the **When output file exists** drop-down menu, select **Overwrite** to substitute files with the same file names.
2. **Delimiter** sets the separator symbol between columns. The delimiter characters available are comma, semicolon, tab, pipe, and space. Choose **Other** to specify a different character.
3. **Qualifier** sets the wrapper symbol type.
4. Choose the **Encoding Type** to apply to the CSV file. The following options are available:
 - Code Page
 - UTF-8
 - UTF-16LE
 - UTF-16BE
5. Set the Code Page to apply to the CSV file. The following options are available:

• OEM United States	• French Canadian (DOS)	• Greek (Windows)
• Greek (DOS)	• Nordic (DOS)	• Turkish (Windows)
• Baltic (DOS)	• Cyrillic (DOS)	• Baltic (Windows)
• Western European (DOS)	• Greek, Modern (DOS)	• Cyrillic (KOI8-R)
• Central European (DOS)	• Japanese (Shift-JIS)	• Cyrillic (KOI8-U)
• OEM Cyrillic	• Central European (Windows)	• Western European (ISO)
• Turkish (DOS)	• Cyrillic (Windows)	• Central European (ISO)
• Portuguese (DOS)	• Western European (Windows)	• Latin 3 (ISO)
• Icelandic (DOS)		• Baltic (ISO)
		• Cyrillic (ISO)

- Greek (ISO)
- Estonian (ISO)
- Japanese (JIS)
- Turkish (ISO)
- Latin 9 (ISO)
- Japanese (EUC)

6. Select the **Include Column Header** check box to include headers in your file.
7. Click **Summary Export** to specify the summary that you want to export.

In the **Summary** dialog box that displays,

- select the summary to export from the **Summary** drop-down menu,
- select the active measure to apply to the summary from the **Active Measure** drop-down menu,
- select the drill level to apply to the summary from the **Drill Level** drop-down menu.

NOTE

Drilling up and drilling down is to collapse and expand a summary data set, respectively.

A lower drill level (e.g., Level 1) results in fewer columns in the exported table.

Click **Ok**.

Additional options for Fixed Text file types

1. In the **When output file exists** drop-down menu, select **Overwrite** to substitute files with the same file names.
2. Click **Column sizes** to adjust the sizes of the columns in your file.

Additional options for Microsoft Excel file types

1. In the **When output file exists** drop-down menu, select **Overwrite** to substitute files with the same file names.
2. Enter your **Table Name**.

NOTE

Click the **Existing Tables** button () to display the names of the tables in the export file when exporting to a file that already exists. Select the desired table name.

3. In the **When table exists** drop-down menu,
 - select **Overwrite** to substitute the data table,
 - select **Append** to append to new rows,
 - select **Update** to update existing rows,
 - select **Update and append** to update existing rows and append new ones.
4. When **Overwrite, Update, Update and append, or Skip** is selected, select the Suppress column header row check box if you prefer to hide the column headers in the exported Excel file.
5. Click **Summary Export** to specify the summary that you want to export.

NOTE

If **Summary Export** is selected, mapping and the **Do not match column names** checkbox are not available.

In the **Summary** dialog box that displays,

- select the summary to export from the **Summary** drop-down menu,
- select the active measure to apply to the summary from the **Active Measure** drop-down menu,
- select the drill level to apply to the summary from the **Drill Level** drop-down menu.

NOTE Drilling up and drilling down is to collapse and expand a summary data set, respectively.

A lower drill level (e.g., Level 1) results in fewer columns in the exported table.

Click **Ok**.

6. When **Append** is selected, modify the following:
 - **Suppress column header row:** Column headers in the exported Excel file are hidden.
 - **Do not match column names:** Columns from potentially different tables are left aligned regardless of column names or data types.

Additional options for Microsoft Access file types

1. In the **When output file exists** drop-down menu, select **Overwrite** to substitute files with the same file names.
2. Enter your **Table Name**.

NOTE Click the **Existing Tables** button () to display the names of the tables in the export file when exporting to a file that already exists. Select the desired table name.

3. In the **When table exists** drop-down menu,
 - select **Overwrite** to substitute the data table,
 - select **Update** to update existing rows,
 - select **Update and append** to update existing rows and append new ones.
4. When **Update** or **Update and append** is selected, modify the following:
 - **Column Mapping:** Maps the export column to the destination column.
 - **When update match not found:** Select the **Export to delimited text file** check box to export the unmatched record(s) to a delimited text file when updating existing rows in a table if a matching row is not found in the destination table.
5. When **Overwrite**, **Append**, or **Skip** is selected, modify the following:
 - **Export all:** Exports all the columns in the export table.
 - **Column Mapping:** Maps the export column to the destination column.

Additional options for Altair Panopticon Designer file types

1. In the **When output file exists** drop-down menu, select **Overwrite** to substitute files with the same file names.
2. Enter your **Table Name**.

NOTE

Click the **Existing Tables** button () to display the names of the tables in the export file when exporting to a file that already exists. Select the desired table name.

3. In the **When table exists** drop-down menu,
 - select **Overwrite** to substitute the data table,
 - select **Update** to update existing rows,
 - select **Update and append** to update existing rows and append new ones.

Additional options for Qlik file types

1. In the **When output file exists** drop-down menu, select **Overwrite** to substitute files with the same file names.

Additional options for Tableau file types

1. In the **When output file exists** drop-down menu, select **Overwrite** to substitute files with the same file names.

NOTE

Export to Tableau is performed in .tde format.

2. To publish the Tableau export file to Tableau Server, click the **Publish to Tableau Server** check box under the **Publish to Tableau Server options**. This displays the following Tableau Server settings.
 - **Data Source Name:** The name of the Data Source on the Tableau Server.
 - **Site:** The address of the data source.
 - **Path to TABCMD:** The route to the executable file of the TableauServerTabcmd application.
 - **Server Address:** The address of the Tableau Server.
 - **Username:** The login name for the Tableau Server.
 - **Use Password File:** Allows the use of .txt file with the password.
 - **Password:** The password to the Tableau Server (if the option Use Password File is enabled, it allows to specify the path to the Password File).
 - **Proxy Address:** The address of the proxy server (in HOST:PORT format).
 - **Publish Type:** Allows to specify an action for the Data Source:
 - ◆ **Overwrite Data Source:** Overwrites the data in the Data Source with the data from the export table (date modified is changed);
 - ◆ **Replace Data:** Replaces the data in the Data Source with the data from the export table (date modified is not changed);
 - ◆ **Append Data:** Adds data from the export table to the end of the Data Source table.

Additional options for OleDb and ODBC file types

1. Type in the **Connection String** or select it via the **Data Link Wizard** with all the required data.
2. Select the **Include password** check box to input a password.
3. Select the **Bulk Behavior** check box to enhance export performance.

NOTE This option is applicable for Oracle Database and Microsoft SQL Server OLEDB providers only.

4. Select the **Use Transaction** check box to revert the export via rollback if an error occurs during the export.

NOTE The **Use Transaction** check box is only available when the **Bulk Behavior** check box is selected.

Selecting this option will affect the performance of the table export and may require a large amount of system resources, especially for the target destination. Therefore, make sure to test this option first and then make the necessary adjustments.

The transaction is valid for the export operation only and not for the overall process.

5. Enter your **Table Name**.
6. In the **When table exists** drop-down menu,
 - select **Overwrite** to substitute the data table,
 - select **Update** to update existing rows,
 - select **Update and append** to update existing rows and append new ones.
7. When **Update** or **Update and append** is selected, modify the following:
 - Click **Column Mapping** to map the export column to the destination column.
 - **When update match not found**: Select the **Export to delimited text file** check box to export the unmatched record(s) to a delimited text file when updating existing rows in a table if a matching row is not found in the destination table.
8. When **Overwrite**, **Append**, or **Skip** is selected, modify the following:
 - **Export all**: Exports all the columns in the export table.
 - Click **Column Mapping** to map the export column to the destination column.

Additional options for Microsoft Power BI file types

1. Enter your **Table Name**.
2. Select a value from the **When table exists** drop-down list.
3. Click the **Set Power BI Login Data** button to open the Power BI Login form.

NOTE If the data on logged in user has been already received, then the button name becomes **Update Power BI Login Data**.

Additional options for IBM Cognos Analytics file types

1. Enter your **Table Name**.
2. Select some value from the **When table exists** drop-down list.

3. Type the value of the Cognos Analytics server URL to the **Server Url** field.
4. Click the **Test** button to get all namespaces.
5. Select the corresponding **Namespace**.
6. Type the correct **Username** and **Password**.
7. Click the **Check Credentials** button to validate credentials.

NOTE If you want to log in anonymously, select the **Is Anonymous** check box.

Additional options for SAS Transport Format file types

1. In the **When output file exists** drop-down menu, select **Overwrite** to substitute files with the same file names.
2. Enter your **Table Name**.

NOTE Click the **Existing Tables** button () to display the names of the tables in the export file when exporting to a file that already exists. Select the desired table name.

The table name is limited to 8 characters in length.

Additional options for JSON file types

1. In the **When table exists** drop-down menu,
 - select **Overwrite** to substitute files with the same file names,
 - select **Add data** to add the new data to the existing one,
 - select **Skip** to stop the export operation from overwriting an existing data.

Automator supports the following plug-ins

Datawatch.DataPrep.Engine

- PDF Report
- Text Report
- Delimited Text
- JSON
- XML
- HTML
- Microsoft Access
- Microsoft Excel
- ODBC

Panopticon.DataDirect

- DB2
- Hadoop Hive

- Cloudera Impala
- Informix
- MonogoDB
- MySql
- Oracle
- PostgreSQL
- Amazon Redshift
- Salesforce
- SqlServer
- SybaseIQ
- Teradata

Panopticon.CloudantPlugin

- IBM Cloudant

Panopticon.DatawatchPlugin

- Monarch Server – Report Warehouse

Panopticon.ODataPlugin

- Odata

Panopticon.SplunkPlugin

- Splunk

Panopticon.BusinessObjectsUniversePlugin

- Business Objects Universe

Panopticon.GoogleAnalyticsPlugin

- Google Analytics

Export Elements

Export elements in the Process Designer tool are sets of instructions how the parsed data is exported by Automator.

The table below specifies functionality of the **Export** group elements.

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
Hub	Defines in what way multiple input files should be grouped and the order in which multiple input files will be processed	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> File inputs processing: <ul style="list-style-type: none"> • Group — all input files are processed in a group • Individually (Export Only) — each input file is processed separately by the following element types: Report Export, Table Export, Summary Export, DataPrep Export, Verify Model, and Verify Workspace • Individually – each input file is

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		<p>processed by all elements after the Hub separately</p> <p>NOTE: When the Individually option is selected, the Process individual inputs in parallel check box is shown. Once this check box is selected, the icon of the Hub element changes from  to . This is because all actions after the Hub element are performed in parallel, meaning all actions are started consecutively and completed consecutively.</p> <p>The same change can be seen for the export elements when the Individually (Export Only) option is selected. For example,  becomes .</p> <p>As for the Individually option, the same change in the appearance of icons is observed not only in the export elements but also in the distribution and script elements.</p> <p>Note that only 10 inputs can be processed in parallel under the Individually setting because the default Max Threads setting in the Automator Dispatcher Service Host is 10. This value can be changed in Configurator. For more details about the Max Threads setting, refer to the Configurator Guide in the Monarch Server Installation Guide (see Configuring Servers).</p> <ul style="list-style-type: none"> <input type="checkbox"/> Order in which to process multiple files: <ul style="list-style-type: none"> • No order — input files are processed in random order • Alphabetical — input files are processed alphabetically according to their names • Old First — input files are processed in chronological order, i.e., from oldest files to newest ones • New First — input files are processed in reverse chronological order, i.e., from newest files to oldest ones
Report Export	Exports data to a report (text, PRF, or PDF format)	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Path — path to the required export file location. <p>NOTE: Specifying a .PRF or .PDF extension will generate the corresponding file type accordingly. Specifying .TXT or any other extension will generate a file with text-based content only.</p> <p>Actions to select export file path:</p>

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		<ol style="list-style-type: none"> Click the Browse button to open the Browse dialog. Select the data source type in the Source Type drop-down list (currently, Local, Web, or Named Paths). Depending on the selected data source type, specify the other parameters: <ul style="list-style-type: none"> For the Local type: Location of the export file and Path to this file using the suggested folder directory; For the Web type: Address which is a URL of the export file; For the Named Paths type: Named Paths and path to the export file using the Named Path Content box (use search capabilities if needed). Click the OK button. <p>NOTE: You may also use naming macros to name the exported files and destination folder.</p> <p>To do this, click the Insert Macro icon () and select the naming macros from the drop-down list.</p> <ul style="list-style-type: none"> <input type="checkbox"/> When output file exists — if a file of the specified name and type already exists in the export location, select one of the following options: <ul style="list-style-type: none"> Overwrite — overwrites the data of the existing output file with the data in the current export file. Add — adds the export file data to the end of the specified file. Skip — is used for Automator project exports, this allows you to create a project export which will skip the export operation if a file of the same name already exists. This is useful for automated operations, where you may not want to overwrite a file that already exists. <p>If the export file has PRF or PDF file type, the ADVANCED OPTIONS appear.</p> <p>Detailed information on the Advanced options (both for PRF and PDF) is specified below this table.</p>
Table Export	Exports table data	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Filter — filter to be applied to the table. You can select the All filters option to apply all of the available filters or the No filter option to apply none of them. If you choose the All filters option, you also need to

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		<p>specify the Automatic Naming options:</p> <ul style="list-style-type: none"> • By files — produces a file for each filter (set as default) • By tables — produces a table for each filter <p><input type="checkbox"/> Sort — sorting order to the data you are exporting, it is selected from the drop-down list.</p> <p><input type="checkbox"/> Output type — defines how the output will be produced, includes two options:</p> <ul style="list-style-type: none"> • File system • Database <p>NOTE: If you select File system, you must specify File options. If you select Database, you must specify Connection options. See description of Connections options and File options below this table.</p> <p>Detailed information on the Connection options and File options is specified below this table.</p>
Summary Export	Exports summary data	<p>NOTE: Settings of the Summary Report element are disable if the model is not specified or does not have any summary.</p> <ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Summary — summary to be exported. You can select the All summaries option to export all of the available summaries. If you choose the All summaries option, you also need to specify the Automatic Naming option: <ul style="list-style-type: none"> • By files — when exporting multiple summaries, a file is exported for each summary, and the files are named according to the summary names. • By tables — when exporting multiple summaries, a table is produced in the export file for each summary. <input type="checkbox"/> Drill State — you can select the following drill states: <ul style="list-style-type: none"> • Most detailed — exports the summary in the most detailed (i.e., drilled down) drill state. • Current — exports the summary in the current drill state. <input type="checkbox"/> Measure — is used to export summary that contains multiple measures. You can select the following options: <ul style="list-style-type: none"> • Active measure or measure grouping — exports only the current

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		<p>measure or measure grouping.</p> <ul style="list-style-type: none"> • All measures as separate exports — exports all of the measures as separate exports. <p>NOTE: If you choose this option, specify the desired Automatic Naming setting, either By files or By tables.</p> <ul style="list-style-type: none"> <input type="checkbox"/> Split — select the Each distinct value of the first key makes a separate file or table check box to have Automator split the summary export according to each value change in the first key (i.e., the leftmost one). <p>NOTE: If you choose this option, specify the desired Automatic Naming setting, either By files to have each distinct value of the first key make a separate file, or By tables to have each change of it make a separate table.</p> <ul style="list-style-type: none"> <input type="checkbox"/> Filter — filter to be applied to the summary. You can select the All filters option to apply all of the available filters or the No filter option to apply none of them. If you choose the All filters option, you also need to specify the Automatic Naming option: <ul style="list-style-type: none"> • By files — produces a file for each filter (set as default) • By tables — produces a table for each filter <input type="checkbox"/> Sort — sorting order to the data you are exporting, it is selected from the drop-down list. <input type="checkbox"/> Output type — defines how the output will be produced. You can select the following options: <ul style="list-style-type: none"> • File system • Database <p>NOTE: If you select File system, you must specify File options. If you select Database, you must specify Connection options.</p> <p>Detailed information on the Connection options and File options is specified below this table.</p>

Advanced options

Advanced options for PRF

- Encryption:** Password-protects the portable report.
 - Select the **Apply encryption** check box.

- Press the **Change...** button and specify a password for the PRF via the Change PRF Password dialog.
- Compression:** Select one of the following compression options:
 - **None:** Specifies no compression of the portable report contents.
 - **Medium (Fast):** Compresses and decompresses slightly faster than the Maximum compression option, and yields a compression ratio of approximately 5:1. This compression ratio may vary depending upon the density of information in your reports.
 - **Maximum:** Compresses and decompresses slightly slower than the Medium compression option but typically yields a compression ratio of 10:1. This compression ratio may vary depending upon the density of information in your reports.

NOTE If you plan to distribute the portable report you should compress it by selecting the Medium or the Maximum option.

Connection options

- Connection String**
 - Enter the connection string manually, or
 - click the **Browse for ...** button and select **Data Link Wizard** to open the Data Link Wizard window. For more information, see Connection Type Wizard.
 - click the **Browse for ...** button and select **Connection Definitions** to open the Connection Definitions window and select from the existing connection definitions. For more information, see Connection Definitions.

After you made your choice, the **Connection string** field displays the connection string for the data you want to connect to.

- Select the **Include Password** check box if your connection requires password. For security reasons, the password is displayed here.

NOTE Passwords are saved in an encrypted format using 1024-bit encryption strength.

- Select the **Bulk Behavior** check box to enhance table export performance.

NOTE This option is applicable for Oracle Database and Microsoft SQL Server OLEDB providers only.

- Select the **Use Transaction** check box to revert the export via rollback if an error occurs during the export.

NOTE The **Use Transaction** check box is only available when the **Bulk Behavior** check box is selected.

Selecting this option will affect the performance of the table export and may require a large amount of system resources, especially for the target destination. Therefore,

make sure to test this option first and then make the necessary adjustments.

The transaction is valid for the export operation only and not for the overall process.

- ❑ **Test Connection String:** Tests whether or not the data link was successful (i.e., to test whether you can connect to the specified OLE DB or ODBC provider). If the connection is successful, the **Test Connection Succeeded** message is displayed.

File options

- ❑ Destination
 - Path to the desired export file if File-auto-name by is None.
 - Path to the desired export folder if File-auto-name by is Filter, Summary, Measure titles or The first key value.

To select Destination:

- a. Click the **Browse** button to open the Browse dialog.
 - b. In the Source Type list, select the data source type from the following: **Local, Web or Named Paths**.
 - c. Depending on the data source type specify other parameters:
 - ◆ **Local:** Specify export file location and path to it;
 - ◆ **Web:** Specify export file address;
 - ◆ **Named Paths:** Select the appropriate named path and select the export file path from the list of displayed in the Named Path Content box.
 - d. Click the **OK** button.
- ❑ File auto-name by
 - **None:** Select if you want export produces only one file;
 - **Summary:** Select if you want export produces a file for each summary (this option available only if “All Summaries” filter is selected);
 - **Measure titles:** Select if you want export produces a file for each measure title (this option available only if “All measures as separate exports” filter is selected);
 - **The first key value:** Select if you want export produces a file for each key value (this option available only if “Each distinct value of the first key makes a separate file or table” filter is selected);
 - **Filter:** Select if you want export produces a file for each filter (this option available only if “All Filters” filter is selected).
 - ❑ **File naming:** Name of the desired export file (available only if File-auto-name by is Filter, Summary, Measure titles or The first key value).

NOTE You may also use the naming macros to name the exported files and destination folder. To do this, click the **Insert Macro** icon () and select the naming macros from the drop-down list.

- ❑ **Export file type:** Select type of export file.
The following options are available:
 - Adobe PDF (*.pdf)
 - Panopticon Designer Data Source (*.dwx)

- dBase (*.dbf)
- Delimited Text
- Fixed Text
- Microsoft Access ACCDB (*.accdb)
- Microsoft Access MDB (*.mdb)
- Microsoft Excel XLS (*.xls)
- Microsoft Excel XLSX (*.xlsm)
- Microsoft Excel XLSX (*.xlsx)
- QlikView QVX (*.qvx)
- SAS Transport File Format V5 (*.xpt)
- Tableau Data (*.csv)
- Tableau Data Extract (*.tde)
- Web File (*.htm)
- Web File (*.html)
- XML File (*.xml)

NOTE

When the dBase, Microsoft Access ACCDB, or Microsoft Access MDB export file type is selected, the **Export Behavior** drop-down menu becomes available. The available options are:

- **Default** - Use this option for all new processes.
- **ACE DAO** - Use this option for better performance for large datasets. All exports in the system with this behavior will be executed concurrently. ACE OleDb will be used as fallback when output tables exist option is "Update existing rows" or "Update existing rows and append new."
- **ACE OleDb** - Use this option for better reliability. This behavior is slower than ACE DAO, but it allows to handle exports in parallel.

- ❑ **When output file exists:** If a file of the specified name and type already exists in the export location, select one of the following options:
 - **Overwrite:** Overwrites the data of the existing output file with the data in the current export file.
 - **Add:** Adds the export file data to the end of the specified file.
 - **Skip:** Used for Automator project exports, this allows you to create a project export which will skip the export operation if a file of the same name already exists. This is useful for automated operations, where you may not want to overwrite a file that already exists.

Table options

- ❑ **Table auto-name by:**
 - **None:** Select if you want export produces only one table;
 - **Summary:** Select if you want export produces a table for each summary (this option available only if All Summaries filter is selected);
 - **Measure titles:** Select if you want export produces a table for each measure title (this option available only if the All measures as separate exports filter is selected);

- **The first key value:** Select if you want export produces a table for each key value (this option available only if the Each distinct value of the first key makes a separate file or table filter is selected);
 - **Filter:** Select if you want export produces a table for each filter (this option available only if the All Filters filter is selected).
- Table name:** Displays the current name of the table. You can assign a new table name by entering it here.
- Existing Tables:** Press the **Existing Tables** button () to display the names of the tables in that file when exporting to a file that already exists.
- When table exists options:
- **Overwrite:** Overwrites the data in the existing table with the data in the export table.
 - **Append:** Adds the export table data to the end of the selected existing table.
 - **Update:** Updates the existing rows in a table.
 - **Update and append:** Updates existing rows in a table and appends new rows to it.

NOTE In the **When update match not found** setting, click the **Export to delimited text file** check box to export the unmatched record(s) to a delimited text file when updating existing rows in a table, particularly if a matching row is not found in the destination table.

- **Skip:** Skips the export operation if a table of the same name already exists. This is useful for automated operations, where you may not want to overwrite a table that already exists.

PDF options (displayed of export file is PDF)

- Open Password options:
- **Apply open password:** Password-protects the PDF file.
 - **Change...:** Displays the Change PDF Export Open Password dialog. Use this dialog to specify the password that must be supplied to open the PDF file.
- Permissions options:
- These options let you restrict the permissions available to users of the PDF file.
- **Apply permissions:** Specifies permission options for the PDF file.
 - **Change...:** Displays the Change PDF Export Permission Password dialog, with which you can specify a password for the exported PDF file(s).
 - **Printing allowed:** Allows the PDF file to be printed.
 - **Content extraction allowed:** Allows copying (e.g., cut and paste) of the PDF file's content.
 - **Editing allowed:** Allows editing of the PDF file.
 - **Annotation (comments) allowed:** Allows comments to be added to the PDF file.

NOTE If you do not specify PDF file permissions, users will have full access to it.

- Encryption options:
- Select one of the following encryption options.

NOTE One (or both) of the passwords must be applied to enable encryption.

- **RC4 (128-bit):** Encrypts the PDF file with RC4 encryption. WARNING: This option is not recommended for extremely confidential data. Use the AES option instead.
- **AES:** Encrypts the PDF file with the Advanced Encryption Standard. **NOTE:** If you choose this encryption method, Adobe Acrobat 7 or higher must be used to open the PDF file, as earlier versions of Acrobat do not support AES encryption.

Excel advanced options (displayed of export file is XLS/XLSX)

The following options are available for Summary Exports:

- Embed the Page Setup header in the Worksheet Title:** Click this option to make the Summary page header (defined using the Page Setup dialog) part of title of exported Excel Worksheet.
- Include Outline to Enable Drill Up/Down in Excel:** Click this option to include an outline in the exported Excel worksheet. The outline will allow you to drill up and down in Excel.
- Include Formulas to Facilitate Editing in Excel:** Click this option if you want Automator to attempt to translate subtotal and grand total calculations into Excel formulas in the exported Excel file.
- Conditional Formats implement measure coloring rules rather than as fixed formats...Use conditional formatting if required conditions are fewer than:** This option implements Conditional Formatting Rules in the exported Excel file, as long as the number of rules would not exceed the specified amount. The Conditional Formatting feature in Excel provides a mechanism to modify the display attributes of a cell based upon the value of that cell. Automator leverages this feature by creating conditional formats from the rules specified on the Coloring and Limits dialog of a measure's properties. Depending upon the rules defined for each measure, up to 3 conditional formatting rules can be generated for each value of an exported measure.

The following options are available for XLS Table Exports:

- Filter:**
 - **Add AutoFilter drop down lists to column labels in the Table:** Select this check box to enable Excel's AutoFilter feature so that it is ready and active upon opening the exported Excel file.

The following option is available for XLSX Table Exports:

- Filter:**
 - **Add AutoFilter drop down lists to column labels in the Table:** Select this check box to enable Excel's AutoFilter feature so that it is ready and active upon opening the exported Excel file.
- Pivot:**
 - **Include Pivot Table sheet:** This option is available for Table exports to XLSX files. To modify the pivot settings, click **Pivot Table**.
 - ◆ **Pivot Table Settings dialog box**
 - **Fields tab:** Use this tab to set the desired pivot field filters, columns, rows, and values.
 - **Options tab:** Use this tab to specify the Pivot Table name, corresponding action if the Pivot Table exists, style, and grand total parameters.

Publish to Tableau Server options (displayed of export file is TDE)

This option allows to publish **Tableau export** to **Tableau Server**.

- Publish to Tableau Server:** Select this option to display Tableau Server settings.
- Data Source Name:** The name of the Data Source on the Tableau Server.
- Site:** The address of the data source.

- ❑ **Path to TABCMD:** The route to the executable file of the TableauServerTabcmd application.
- ❑ **Server Address:** The address of the Tableau Server.
- ❑ **Username:** The login name for the Tableau Server.
- ❑ **Use Password File:** Allows the use of .txt file with the password.
- ❑ **Password:** The password to the Tableau Server (if the option Use Password File is enabled, it allows to specify the path to the Password File).
- ❑ **Proxy Address:** The address of the proxy server (in HOST:PORT format).
- ❑ **Publish Type:** Allows to specify an action for the Data Source:
 - **Overwrite Data Source:** Overwrites the data in the Data Source with the data from the export table (date modified is changed);
 - **Replace Data:** Replaces the data in the Data Source with the data from the export table (date modified is not changed);
 - **Append Data:** Adds data from the export table to the end of the Data Source table.

Prerequisites Elements

Prerequisites elements in the Process Designer tool check prerequisites specified by the user.

The table below specifies functionality of the **Prerequisites** group elements.

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
Check File	<p>File prerequisite that tests for existence of the specified file or files. This can be a file name or a wildcard specification for the file.</p> <p>When the process runs, Automator will check the specified prerequisites to see that they have been met, and if they have been, the operations defined by grey connectors will run.</p> <p>If they have not, the operations defined by red connectors will run.</p>	<ul style="list-style-type: none"> • Name — name of the graphic element. • File Path — path that shows the File Prerequisite location. <p>The File Path parameter can be filled out manually or using Browse.</p> <p>Actions:</p> <ol style="list-style-type: none"> 1. Click the Browse button to open the Browse dialog. 2. Select the data source type in the Source Type drop-down list (currently, Local or Named Paths). 3. Depending on the selected data source type, specify the other parameters: <ul style="list-style-type: none"> • For the Local type: Location of the prerequisite and Path to this file using the suggested folder directory; • For the Named Paths type: Named Paths and the path to the prerequisite using the Named Path Content box (use search capabilities if needed). • Click the OK button. • Test — use this button to check file availability in the specified location.
Check Cmd	Command line prerequisite that	<ul style="list-style-type: none"> • Name — name of the graphic

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
	<p>runs the specified command line and tests that it returns the given success code.</p> <p>When evaluating this prerequisite, the server runs the command in a hidden window and checks that its return code matches the expected success code.</p> <p>By default the success code is 0 (zero), which is the default success code for programs if they complete successfully.</p>	<p>element.</p> <ul style="list-style-type: none"> • File Path — path that shows the Command Line Prerequisite location. <p>The File Path parameter can be filled out manually or using Browse.</p> <p>Actions:</p> <ol style="list-style-type: none"> 1. Click the Browse button to open the Browse dialog. 2. Select the data source type in the Source Type drop-down list (currently, Local or Named Paths). 3. Depending on the selected data source type, specify the other parameters: <ul style="list-style-type: none"> • For the Local type: Location of the Command Line Prerequisite and Path to this file using the suggested folder directory; • For the Named Paths type: Named Paths and the path to the Command Line Prerequisite using the Named Path Content box (use search capabilities if needed). • Click the OK button. • Arguments — arguments for command line. • Success code — integer code which returns by the specified command line. • Test — use this button to check returning the command line success code.
Check Input	Input prerequisite that tests for the existence of the specified input.	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> All files must exist — specifies that all of the input files should exist. When the process runs, Automator will check all specified inputs to see that they have been met, and if they have been, the operations defined by grey connectors will run. If they have not, the operations defined by red connectors will run. <input type="checkbox"/> At least one file must exist — specifies that at least one of the input files should exist. When the process runs, Automator will check the specified inputs to see that they have been met according to the Check Input settings, and if at least one of them has been, the operations defined by grey connectors will run. If they have not, the operations defined by red connectors will run. <input type="checkbox"/> Check file consistency — specifies that

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		<p>a file size check be carried out once the prerequisite is found. This allows the system to cope with slow, unreliable, or multi-stage write sources. Automator will test the input file by ensuring that the file size has not changed for a specified period and that the file is fully ready to be processed.</p> <ul style="list-style-type: none"> <input type="checkbox"/> If Check Input is not added and the specified inputs do not exist, the process execution will not fail because non-existent inputs will be ignored.
Verify Model	<p>Automator supports verification of report inputs on a process-by-process basis.</p> <p>NOTE: Verification is not available for database inputs. If a process calls for database inputs, the verification options are simply ignored.</p> <p>Verification settings allow you to specify the kind of verifications you want it to perform while processing. When verification is enabled, Automator may verify field boundaries, data types and filter results.</p>	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Filter produces at least one row — checks that when the specified filter is applied to the data, the resulting table contains at least one row. <input type="checkbox"/> Filter produces no rows — checks that when the specified filter is applied to the table, no rows are produced. For example, you may have created a filter that performs checks on your data to ensure that it is valid, and if any bad rows are returned by this filter, you wish to abort the procedure. <input type="checkbox"/> Unfiltered table has at least one row — checks that the unfiltered table contains at least one row of data. <input type="checkbox"/> Field boundaries and data types for text inputs — checks that all fields are in the correct location, have correct length, and are of the correct data type. This is equivalent to using the verify feature in Monarch. Note that any verification settings recorded in the Monarch model will be honored. These can be found in the Field List of Monarch. <p>NOTE: This feature is not available for database and PDF inputs.</p> <ul style="list-style-type: none"> <input type="checkbox"/> Text field lengths — checks that all text fields have valid field length. <input type="checkbox"/> Numeric field decimals — checks that all numeric fields have valid decimals. <input type="checkbox"/> Numeric field lengths — checks that all numeric fields have valid field length. <input type="checkbox"/> Database fields — checks that all database fields have been defined correctly. <input type="checkbox"/> Calculated fields — checks that all calculated fields have been defined correctly. <input type="checkbox"/> Date/time format precisions — checks

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		<p>that all date/time fields have the correct format precision.</p> <p>When you select a verification option, verification is activated.</p> <p>If verification fails setting</p> <ul style="list-style-type: none"> <input type="checkbox"/> Continue. Automator will continue processing the associated actions for all inputs even if a verification failure of the specified kind is encountered. <input type="checkbox"/> Continue with verified only. For input files processed as a group, Automator will continue processing the associated actions for these inputs only if they were all verified successfully. For input files processed individually, Automator will continue processing the associated actions only for the inputs verified successfully; those that failed verification will not be processed. <input type="checkbox"/> Stop. The first failed verification causes the Verify Model to immediately fail. <input type="checkbox"/> Stop without error notifications. The job is stopped, but no email alert is sent. In this case, the job log message is "Job completed without errors." <p>NOTE: To use input files that were not verified successfully, the Verify Model element should be connected to another element using the green line.</p>
Verify Workspace	Automator allows results in a workspace to be verified so that users can stop, proceed, or proceed with notification.	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Load Plans to be verified — select the load plan/s for verification. <p>The user can select several load plans by ticking their corresponding checkboxes. To export all load plans, select the All Load Plans checkbox.</p> <p>Verification options — choose the verification to be done:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Verify DataSource availability – select this option to check whether the data source specified is available. <input type="checkbox"/> Check that number of rows is – select this option to check if the number of rows meets the specified value: <ul style="list-style-type: none"> • Greater than or equal to: set the minimum number of rows for verification • Equal to zero: set the number of rows for verification to zero <p>If verification fails setting</p>

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		<ul style="list-style-type: none"> <input type="checkbox"/> Continue. Automator will continue processing the associated actions for the workspace even if a verification failure of the specified kind is encountered. <input type="checkbox"/> Continue with verified only. Automator will continue processing the associated actions for the workspace only if it was verified successfully. <input type="checkbox"/> Stop. The first failed verification causes the Verify Workspace to immediately fail. <input type="checkbox"/> Stop without error notifications. The job is stopped, but no email alert is sent.

Misc Elements

Misc elements in the Process Designer tool enhance capabilities of the Automator.

The table below specifies functionality of the **Misc** group elements.

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
Distribution	<p>Distributions specify actions, called distribution actions, that distribute or handle files and information outside of Automator.</p> <p>Distributions can be specified for input files, export files, and portable reports, and can be configured with the Distribution Editor and the Script Editor. Distributions can be used for:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Sending emails (with or without attachments) <input type="checkbox"/> RSS feeds <input type="checkbox"/> Moving, copying, and deleting files (to both the file system and to FTP/SFTP-compliant and WebDAV-compliant servers using http) <input type="checkbox"/> Writing to the Windows Event Log 	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <p>Acceptable distribution actions:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Log — allows you to create an event written to WindowsApplication Event Log. <input type="checkbox"/> Email — allows you to email files to another user <input type="checkbox"/> Move — allows you to move files to another location, either to the file system using drive letter or UNC naming, or to FTP/SFTP and HTTP locations <input type="checkbox"/> Copy — allows you to copy files to another location, either to the file system using drive letter or UNC naming, or to FTP/SFTP and HTTP locations <input type="checkbox"/> Delete — allows you to delete all files specified by the distribution <input type="checkbox"/> RSS — allows you to "pull" information as opposed to the "push" model of Email <input type="checkbox"/> Email By Key — allows you to set the recipients of the E-mail, based on key. This option available only for the Summary Export element. <input type="checkbox"/> Export to Amazon S3 – allows

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		<p>you to export input and export files to Amazon S3</p> <p>Detailed information on distribution settings is specified in the Distribution Settings table below.</p>
Script	<p>Automator supports the use of Visual Basic.NET script.</p> <p>You can find collection of Visual Process Designer scripts is the Process Designer Scripting API section.</p>	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Script field — displays the current script. <input type="checkbox"/> Run – runs the current script <input type="checkbox"/> Insert Script — allows to quickly generate VB.NET script for frequently used tasks. Click to select the necessary options. For details, refer to Inserting a Script section. <input type="checkbox"/> User Scripting Help — opens Help on user scripting. <p>If the scripts have errors, all the errors are displayed below the Script field.</p> <p>All namespaces and references, which specified in the Global Scripts of Standard Processes, are imported in scripts of Visual Processes.</p> <p>There is no limit to what you can do with the scripting function of Visual Process Designer Scripting API. The entire Microsoft .NET Framework is available for use.</p> <p>You can use Visual Studio.NET to help create your scripts, or you can obtain the free Microsoft .NET Framework SDK.</p> <p>Note that while you can compile and test your scripts with the Visual Process Script, you must ensure that you are not using message boxes or other functions that interact with the UI when you are running the scripts as part of a process.</p> <p>NOTE: Automator users can only edit scripts if they are granted editing privileges by the Admin.</p>
Credentials	<p>Use it to specify the HTTP, FTP/SFTP, and Amazon S3 login credentials for input, model, export files from an FTP/SFTP site, a SharePoint repository, or Amazon S3 storage.</p>	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> HTTP Credentials — specify a name in the User name field and password in the Password field. <p>NOTE: To use the current windows login, select the corresponding check</p>

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		box. <ul style="list-style-type: none"> <input type="checkbox"/> FTP/SFTP Credentials — specify a name in the User name field and password in the Password field. <input type="checkbox"/> Amazon S3 Credentials — specify an ID in the Access key ID field and an access key in the Secret access key field.
Timer	<p>When configuring a process, one of the settings you can specify is a process retries setting.</p> <p>This setting allows a process workflow to be retried if it fails for some reason, such as due to a process prerequisite not being met.</p> <p>When specifying a process retries setting, you may specify the interval for the retry in hours and minutes, as well as the number of retries to attempt.</p> <p>NOTE: The Timer object can also be configured such that it sends an alert about failures in the process after the set number of retries has been exhausted. For example, once the number of retries has been exceeded, the Timer object could trigger an email distribution informing staff of the failed process.</p>	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Retry Type: select one of the following options: <ul style="list-style-type: none"> • Indefinitely: Select this option to continue retrying the process indefinitely until it runs successfully. • Retry at most: Select this option to enter the maximum number of times you want the Automator to retry running the process. <input type="checkbox"/> Retry every — the desired retry interval in hours, minutes or seconds.
Visual Process	<p>This functionality allows processes to be run in a chain.</p> <p>Example:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Add two processes. <input type="checkbox"/> Connect the two processes. <input type="checkbox"/> Run the new process. <p>In this case, the first process starts to run. As soon as it is completed, the second process starts. If the first process fails, the second process does not start, except if the two processes are connected by a red connector.</p>	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the visual process <input type="checkbox"/> Process name field — click the Browse button to select a saved visual process <input type="checkbox"/> Description — description of selected visual process <input type="checkbox"/> Preview Process — click this option to preview the selected visual process
Standard process	<p>This functionality allows processes to be run in a chain.</p> <p>Example:</p> <ul style="list-style-type: none"> <input type="checkbox"/> Add two processes. <input type="checkbox"/> Connect the two processes. <input type="checkbox"/> Run the new process. <p>In this case, the first process starts to run. As soon as it is completed, the second process starts. If the first process fails, the second process does not start, except if the</p>	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the visual process <input type="checkbox"/> Process name field — click the Browse button to select a standard process <input type="checkbox"/> Description — description of the selected standard process <input type="checkbox"/> Projects — list of projects connected to the standard process; click the project name to view its information

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
	two processes are connected by a red connector.	
File Burster	This functionality allows file or web inputs to be split into smaller files according to defined properties.	<ul style="list-style-type: none"> <li data-bbox="911 363 1333 420">❑ Input – select or multiple file inputs or web inputs to be bursted <li data-bbox="911 432 1333 541">NOTE: Only ASCII files are supported as input for file burster item (e.g., .txt and .prn inputs are supported but .pdf inputs are not). <li data-bbox="911 558 1333 772">❑ Recognize Line Key, Offset – optional identifier within the input files to be bursted; bursting proceeds when the key is recognized within the input file; <i>offset</i> specifies the number of characters to be counted from the left of the key <li data-bbox="911 793 1333 982">❑ Trim leading characters – this option is available when Defined Bursting is selected; specify the number of characters to be trimmed from each line throughout the output document (small values are usually used, e.g., 1 or 2) <li data-bbox="911 1003 1333 1150">❑ Burst Type (Defined or Pages) <ul style="list-style-type: none"> <li data-bbox="959 1045 1333 1150">Defined: Used when the input file contains a table. The table is parsed according to the following definitions: <ul style="list-style-type: none"> <li data-bbox="959 1171 1333 1276">• Index Table Start Maker, Offset – an indicator that marks where in the input file the index table begins <li data-bbox="959 1297 1333 1381">• Line Validator, Offset – specifies the symbols that mark the beginning of a line <li data-bbox="959 1402 1333 1591">• Line Field Offset, Line Field Length – specifies the column number in the input file that marks the end of a line; field length specifies the number of characters that make up the line field <li data-bbox="959 1612 1333 1822">• ID Field Offset, ID Field Length – specifies the number of characters in the input file that should mark the beginning of the ID field; field length specifies the number of characters that make up the ID field <li data-bbox="959 1843 1333 1896">• Index Table End Marker, Offset – an indicator that

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		<p>marks where in the input file the index table ends</p> <ul style="list-style-type: none"> • Content Start Marker, Offset – an indicator that marks where in the input file the content of the table begins <p>Pages: A Monarch model is used to define the page headers of a file and parse it into smaller files. This is a better option when dealing with large files.</p> <ul style="list-style-type: none"> • Process successive number files as one – select this option if you want consecutively numbered files in one location to be bursted as one file • Model – specify the model file to be used for the input file • Pages per Output File – specify the number of pages that each output file should contain <ul style="list-style-type: none"> <input type="checkbox"/> Destination Path – location of generated files after bursting <input type="checkbox"/> Output Filename Mask – name pattern for the generated files <ul style="list-style-type: none"> • Header Output Mask – this option is available in Defined Bursting; specifies name pattern for the generated file containing table header information <input type="checkbox"/> Error Directory – location of generated files containing errors after bursting <p>NOTE: All burst output files will inherit the “last modified” timestamp of the original parent input report.</p>

Distribution Settings

The table below specifies distribution actions.

DISTRIBUTION ACTION	DESCRIPTION	PROCEDURE AND COMMENTS
Log	Allows you to create an event written to	<input type="checkbox"/> Type: Select the type from the

DISTRIBUTION ACTION	DESCRIPTION	PROCEDURE AND COMMENTS
	<p>WindowsApplication Event Log.</p> <p>This can be useful when using system management systems such as BMC Patrol, Tivoli, or Openview to tie in Automator operations with your overall systems.</p>	<p>drop-down list.</p> <ul style="list-style-type: none"> <input type="checkbox"/> Event ID: Enter the event ID. <input type="checkbox"/> Message: Type the message in the text field. <p>NOTE: You may use the naming macros to provide a message for the log distribution. To do this, click the Insert Macro icon and select the naming macros from the drop-down list.</p> <p>NOTE: When exporting to Knowledge Studio, all three files (*.kdd, *.ksd, and *.ksd-data) are listed individually in the event log.</p>
Email	Allows you to email files to another user	<p>Emailing procedure:</p> <ol style="list-style-type: none"> 1. Specify the recipients' email addresses in the corresponding boxes: From, To, Cc, Bcc. To select recipients from the contacts list, click the Contacts icon. 2. Type the message subject in the Subject box. <p>NOTE: You may use the naming macros to provide a subject line for an email distribution. To do this, click the Insert Macro icon and select the naming macros from the drop-down list.</p> <ol style="list-style-type: none"> 3. Type the message in the Message box. 4. Select the Attach files check box to attach a file for the distribution. 5. Select the Zip check box to archive the attached files. The Zip Options appear. 6. Under Zip Options provide the following information: <ul style="list-style-type: none"> • Specify the Compression level from the following: None, Medium, Maximum. • Select the Store directory path check box if necessary. • Select the Password check box to set the password. 7. Select the Attach links check box to attach a link to file. <p>NOTE: Please specify the Admin Host and Port information in the System Settings.</p> <ol style="list-style-type: none"> 8. Select the One file per mail check

DISTRIBUTION ACTION	DESCRIPTION	PROCEDURE AND COMMENTS
		<p>box to attach only one file per mail.</p> <p>NOTE: When exporting to Knowledge Studio, all three files (*.kdd, *.ksd, and *.ksd-data) are kept together and sent in one email.</p> <p>9. Select the email Priority from the following: Low, Normal, High.</p>
Move	Allows you to move files to another location, either to the file system using drive letter or UNC naming, or to FTP/SFTP and HTTP locations.	<p>Moving procedure:</p> <ol style="list-style-type: none"> In the Destination box, specify the destination folder. You can either enter the destination folder path in the box or select the folder in the Browse section. <ul style="list-style-type: none"> Click the Browse button to open the Browse dialog. From the Named Paths list, select the appropriate named path. Select the folder from the list displayed in the Named Path Content box. Click OK to add the folder. In the Rename as box, specify a new file name if you want to change the file name. <p>NOTE: You may use the naming macros to name the moved files and destination folder. To do this, click the  and select the naming macros from the drop-down list.</p> Select the Change extension to check box to change file's extension. Specify new extension in the enabled text box below. Select the Zip check box, if you want to archive the file. The Zip Options settings appear. <p>NOTE: The Zip check box is disabled if Change extension to check box selected.</p> <p>Under Zip Options, provide the following information:</p> <ul style="list-style-type: none"> Select the Compression level from the following: None, Medium, Maximum. Select the Store directory path check box, if you want to

DISTRIBUTION ACTION	DESCRIPTION	PROCEDURE AND COMMENTS
		<p>store directory path.</p> <ul style="list-style-type: none"> Select the Set password check box to set the password. <p>5. In the When destination files exist option:</p> <ul style="list-style-type: none"> Select Raise error to receive an alert of an existing file. Select Overwrite to overwrite the existing file. Select Skip to skip the existing file. <p>NOTE: When exporting to Knowledge Studio, all three files (*.kdd, *.ksd, and *.ksd-data) are moved and listed in the log individually.</p>
Copy	Allows you to copy files to another location, either to the file system using drive letter or UNC naming, or to FTP and HTTP locations.	<p>Copying procedure is similar to moving procedure (see above).</p> <p>NOTE: When exporting to Knowledge Studio, all three files (*.kdd, *.ksd, and *.ksd-data) are copied and listed in the log individually.</p>
Delete	Allows you to delete all files specified by the distribution.	<p>NOTE: When exporting to Knowledge Studio, all three files (*.kdd, *.ksd, and *.ksd-data) are deleted and listed in the log individually.</p>
RSS	<p>Allows you to "pull" information as opposed to the "push" model of Email.</p> <p>RSS (Really Simple Syndication) is a widely used format for publishing information.</p> <p>The RSS file must reside on a web server and be accessible to users who can read the RSS feed using one of the many free and commercial RSS news readers available on the market, for example, RSSBandit.</p>	<p>1. In the Feed box, specify the RSS file. You can either enter the destination folder path in the box or select the folder in the Browse section.</p> <ul style="list-style-type: none"> Click the Browse button to open the Browse dialog. From the Named Paths list, select the appropriate named path. Select the folder from the list displayed in the Named Path Content box. Click OK to add the folder. <p>2. Select the Create if not found check box if you want to create the file if the specified file is not found.</p> <p>3. In the Title box, enter the item title.</p> <p>4. In the Item description box, enter the item description.</p> <p>NOTE: You may use the naming macros to provide a title and description for an RSS distribution. To do this, click</p>

DISTRIBUTION ACTION	DESCRIPTION	PROCEDURE AND COMMENTS
		<p>the Insert Macro  icon and select the naming macros from the drop-down list.</p> <ol style="list-style-type: none"> In the Link box, enter the name of link to the file. Expand the Channel section to provide the channel parameters: <ul style="list-style-type: none"> Title: Enter channel title. Channel Description: Provide channel description. Link: Enter the name of the link to the file. <p>NOTE: If you leave the Title, Channel Description, and Link fields blank the default values will be automatically assigned.</p> <ul style="list-style-type: none"> Max Items: Specify the maximum number of items. Unlimited: Select this check box if you want to set the unlimited number of items. <p>NOTE: When exporting to Knowledge Studio, all three files (*.kdd, *.ksd, and *.ksd-data) are listed in the RSS file individually.</p>
Email By Key	Allows you to set the recipients of the E-mail, based on key. This option available only for the Summary Export element.	<ol style="list-style-type: none"> From: Type the address or click the Contacts icon to choose a sender. To: Type the address or click the Contacts icon to choose a sender. <p>NOTE: When typing multiple addresses, separate them with semicolon.</p> <ol style="list-style-type: none"> Recipients: Allows you to set the recipients of the e-mail, based on key. Subject: The subject line for the message. You can use the adjacent arrow icon to insert a naming macro. Type the message in the Message box. Select the Attach files check box to attach a file for the distribution. Select the Zip check box to archive the attached files. The Zip Options appear.

DISTRIBUTION ACTION	DESCRIPTION	PROCEDURE AND COMMENTS
		<ol style="list-style-type: none"> 8. Under Zip Options provide the following information: <ul style="list-style-type: none"> • Specify the Compression level from the following: None, Medium, Maximum. • Select the Store directory path check box if necessary. • Select the Password check box to set the password. 9. Select the One file per mail check box to attach only one file per mail. 10. Select the email Priority from the following: Low, Normal, High.
Export to Amazon S3	<p>Allows you to export input and export files to Amazon S3</p> <p>NOTE: This option requires the Credentials element to be connected to the Distribution element. The Credentials element should be set with the appropriate Amazon S3 credentials (i.e., access key ID and secret access key).</p>	<ol style="list-style-type: none"> 1 Specify the URL path of the Amazon S3 storage in the Destination field. 2 Enter a new file name in the Rename as field if you wish to use a new file name for the export/input item. 3 Select the Change extension to check box to change the file's extension. Specify a new extension in the enabled text box. 4 In the When destination files exist option: <ul style="list-style-type: none"> • Select Raise error to receive an alert of an existing file. • Select Overwrite to overwrite the existing file. • Select Skip to skip the existing file. 5 Select the Zip check box if you want to archive the file. The Zip Options settings appear. <p>NOTE: The Zip check box is disabled if Change extension to check box is selected.</p> <p>Under Zip Options, provide the following information:</p> <ul style="list-style-type: none"> • Select the Compression level from the following: None, Medium, Maximum. • Select the Store directory path check box if you want to keep the directory path. • Select the Set password check box to assign a

DISTRIBUTION ACTION	DESCRIPTION	PROCEDURE AND COMMENTS
		password to the archived file.

Output Elements

Output elements are used for integration with other Monarch Server components or other systems / applications. The table below specifies functionality of the **Output** group elements.

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
Hive Output	For loading export data into a Hive table	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Connection String — the string for connecting to the Hive database. <input type="checkbox"/> Table name — name of the Hive table. <input type="checkbox"/> Server — server where the table is stored. <input type="checkbox"/> Directory — directory where the table is stored. <input type="checkbox"/> User Name — name of the creator of the Hive table. <p>Limitation: For this output type, you must use export data in the CSV format.</p>
Report Warehouse Output	For uploading data into a Report Warehouse document type	<ul style="list-style-type: none"> <input type="checkbox"/> Name — name of the graphic element. <input type="checkbox"/> Document type — Report Warehouse data container the report input is uploaded into. <input type="checkbox"/> Document Group — name of the group of document types. <input type="checkbox"/> Processing Type: <ul style="list-style-type: none"> • Read and process internal indexes. This option is selected by default. It is used for reading text and PDF reports where index values are extracted and reports are mined by users. • Process as native document. This option is used for processing any file or document that does not require index value extraction and mining from its content. Native documents can be indexed on fixed values (Index defaults), document date, and filename element values. This option requires licensing. To display native documents, you must install appropriate software on the client's computer (for example, XLS documents require the MS Excel application or another XLS file viewer). <p>ADVANCED OPTIONS</p> <ul style="list-style-type: none"> <input type="checkbox"/> Store Original Report: When reports are

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		<p>read and indexed for storage, their format is changed to enable fast and efficient retrieval and mining. Selecting this option will also store the original report, as read, in the storage system.</p> <ul style="list-style-type: none"> <li data-bbox="813 401 1344 621">❑ Pages / Storage Unit: In storing reports, there are trade-offs between the compression ratio (disk space requirements) and speed of retrieval. A higher number of Pages / Storage Units can provide dramatically increased storage compression but at a cost of reduced retrieval efficiency and performance. <li data-bbox="813 638 1344 831">❑ Pre-mine all processed reports for interactive data analysis: Select this option, if you want table data to be extracted from the reports during the filing process and put into the Monarch Server database. Then the data can be used to build Dynamic View data views. This option requires licensing. <p>Redaction settings are available only if a redaction license is enabled in Monarch Server.</p> <ul style="list-style-type: none"> <li data-bbox="813 942 1344 1052">❑ Store reports redacted: Select this option to allow storage reports to be stored with redaction. In this case reports are stored in the redacted format in storage volume. <li data-bbox="813 1068 1344 1146">❑ Store mining data redacted: Select this option to allow pre-mined data to be stored with redaction. <p>Limitation: For this output type, you can use data from text input or report export.</p>

Process Designer Scripting API

We have provided a short collection of script fragments to take a look at and copy and paste to serve as starting points or fragments of your own scripts.

The Visual Process Designer Scripting API exposes the following members:

- ❑ **AddEvent:** Adds any text to the log (for informational or auditing purposes).
- ❑ **AddInput:** Adds an input item to the process.
- ❑ **AddOutput:** Adds an output item to the items for the specified export.
- ❑ **AddPassword:** Adds a script password.
- ❑ **AddRuntimeFieldValue:** Sets the value for a runtime field. This value will be used, if called for, by subsequent exports.
- ❑ **AddVariable:** Adds a variable to the script.
- ❑ **ClearPasswords:** Clears all script passwords.
- ❑ **ClearRuntimeFieldValues:** Clears values for runtime fields.
- ❑ **ExpandMacros:** Expands macros in a given string, in the context of the entire log.

- ❑ **GetEvents:** Gets the list of events for the log item specified by name.
- ❑ **GetExports:** Gets the list of process export names.
- ❑ **GetInputs:** Gets the list of input items.
- ❑ **GetOutputRowCount:** Gets the row count of Report, Table, Summary, or DataPrep export by export element name; by export element name and location; or by export element name, location, and table name.
- ❑ **GetOutputs:** Gets the list of output items for a process.
- ❑ **GetOutputs (string):** Gets the list of output items for a specified process export.
- ❑ **GetOutputTableNames:** Gets the table names of Report, Table, Summary, or DataPrep export by location.
- ❑ **GetPassword:** Gets the list of script passwords.
- ❑ **GetProcessName:** Returns the name of the process.
- ❑ **GetRuntimeFieldValue:** Returns the current value, if any, for a given runtime field name.
- ❑ **GetTrackingId:** Gets the tracking ID of the current log.
- ❑ **GetVariable:** Gets the list of script variables.
- ❑ **HasPassword:** Returns “true” if the password as added using the AddPassword method
- ❑ **HasRuntimeFieldValue:** Returns true if a value has already been set for a given runtime field name.
- ❑ **RiseAlert:** Rise process or global alert by name with a specified message.

Here are some specific examples.

API.AddInput Method

Adds an input item to the process.

Syntax: *void AddInput(string fileName)*

Parameters:

fileName

Type: System.String

Path to the input file.

API.AddOutput Method

Adds an output item to the items for the specified export.

Syntax: *void AddOutput(string exportName, string location)*

Parameters:

exportName

Type: System.String

Name of process export.

location

Type: System.String

Path to the output file.

API.AddRuntimeFieldValue Method

Adds an output item to the items for the specified export.

Syntax: *void AddRuntimeFieldValue(string fieldName, string fieldValue)*

Parameters:

fieldName

Type: System.String

Name of runtime field.

fieldValue

Type: System.String

Value of runtime field.

API.ClearRuntimeFieldValues Method

Clears values for runtime fields.

Syntax: *void ClearRuntimeFieldValues()*

API.GetExports Method

Gets the list of process export names.

Syntax: *IList<string> GetExports()*

Return Value:

Type: StringCollection

A StringCollection containing the process export names.

API.GetInputs Method

Gets the list of input items.

Syntax: *IList<string> GetInputs()*

Return Value:

Type: StringCollection

A StringCollection containing the process inputs.

API.GetProcessName Method

Returns the name of the process.

Syntax: *string GetProcessName()*

Return Value:

Type: String

The name of the process.

API.GetOutputs Method

Gets the list of output items for a process.

Syntax: *IList<string> GetOutputs()*

Return Value:

Type: StringCollection

A StringCollection containing the process outputs.

API.GetOutputs Method

Gets the list of output items for a specified process export.

Syntax: *ICollection<string> GetOutputs(string exportName)*

Parameters:

exportName

Type: System.String

Name of process export.

Return Value:

Type: StringCollection

A StringCollection containing the outputs of the process export.

API.GetRuntimeFieldValue Method

Returns the current value, if any, for a given runtime field name.

Syntax: *string GetRuntimeFieldValue(string fieldName)*

Parameters:

fieldname

Type: System.String

Name of runtime field.

Return Value:

Type: String

Value for the specified runtime field or empty string.

API.HasRuntimeFieldValue Method

Returns true if a value has already been set for a given runtime field name.

Syntax: *bool HasRuntimeFieldValue(string fieldName)*

Parameters:

fieldname

Type: System.String

Name of runtime field.

Return Value:

Type: Boolean

True if this field has a value already set

Importing a Visual Process

You can import visual processes from the **Processes** page.

You can import a process by selecting the output (in the JSON format) of a previous export process operation.

To import a process

1. On the Navigation Tree, click **Automator** and select **Visual Processes**.
2. On the displayed page, select the **Import Processes** tab.
3. Click the **Upload** button and choose the file to import in the **Open** dialog box. You can select only one file at once.

NOTE You can also import a standard process as a visual process. For this, export the standard processes with the *"Include process dependencies"* option enabled, and then upload them to Visual Processes.

4. Enable the **Change locations on import** check box to substitute all locations values to the selected new location.
5. Enable the **Include process dependencies** check box to import models, workspaces, change lists, and Visual Processes/Standard Processes connected to the processes (if any) to the Server Library.
6. Click the **Import** button to import the processes. A message box informs you that the import operation is complete.

NOTE If you exceed the number of processes allowed by your license, the following message appears.

"The maximum number of processes per your license (XX) will be exceeded with this import and it cannot be completed. You can select at most (NN) processes for import, or you may delete unused processes or contact your Altair sales representative for extended options."

Named Path Mapping

During import of an exported Visual Process that includes named paths, the **Named Path Mapping** window displays. In this window, the named paths in the Visual Process being imported are mapped to the existing named paths in the destination environment. The following options are available:

-  Auto Map Named Path button – Click to automatically map all imported named paths by ID or by Name (depending on the radio button selected). By default, the imported named paths are mapped by Name.
-  Edit Mapping button – Click to manually perform mapping or modify existing mapping.
-  Delete Mapping button – Click to delete currently selected mapping.
-  Delete All Mappings button – Click to delete all listed mappings.

Exporting a Visual Process

You can export visual processes on the **Processes** page.

The output of the export operation is a JSON file (or files) that can then be imported into another Automator Installation.

To export a process

1. On the Navigation Tree, click **Automator** and select **Visual Processes**.
2. On the displayed page, select the **Export Processes** tab.
3. Select the process you want to export. Multiple selection is available.
4. Specify the number of processes per file by selecting one of the following options:

NOTE One process per file if you want each process to be exported to its own JSON file. All processes in one file if you want the exported processes to be exported into one JSON file.

5. Enable the **Include process dependencies** check box to export model, workspace, change list content, and Visual Processes/Standard Processes.
6. Click **Export**.
7. In the displayed dialog box, enter the name of the archived output file.

NOTE If you select *"One process per file"*, the outputs are named <process_name>.json, located in the folder you specify. If you select *"All processes in one file"*, the output uses the name you specify, located in the folder you select.

8. Click **OK** to export the process or **Cancel** to cancel exporting. A message box informs you when the export operation is complete.

Migrating a Standard Process

You can migrate existing standard processes into visual processes.

To migrate a process

1. On the Navigation Tree, click **Automator** and select **Visual Processes**.
2. On the displayed page, select the **Migrate Standard Processes** tab.
3. Select the process you want to export. Multiple selection is available.
4. Enable the **Change locations on import** check box to substitute all locations values to the selected new location.
5. Click the **Migrate** button. A message box will inform you that the migrate operation is complete.

NOTE You can migrate only the processes with projects from the **Server Library** location.

If you exceed the number of processes allowed by your license, the following message appears:

"The maximum number of processes per your license (XX) will be exceeded with this migration and it cannot be completed. You can select at most (NN) processes for migration, or you may delete unused processes or contact your Altair sales

representative for extended options."

Schedules

All schedules for Visual Processes are listed on the **Schedules** page. You can use this page to [add](#), [edit](#) or [delete](#) schedules.

To add a new schedule for a Visual Process

1. On the **Schedules** page under Visual Processes, click . The Schedule page will be displayed.
2. Select the **Suspended** check box if you want to suspend the task.
3. Specify the process to which the schedule will be assigned.
4. Select the task [execution frequency](#) from the following options: **One Time, Hourly, Daily, Weekly, Monthly**.
5. Set the time you want the process to run.
6. Set the start date of the schedule.
7. Select the appropriate timezone from the drop-down list.
8. Click the **Save** button to save the schedule or **Run Now** to start the task immediately.

NOTE

If the Config service is not available and you are using Altair licensing, Automator processes will be switched to "Limp Mode." In this mode, you can run only one process (i.e., one Standard Process and one Visual Process) every hour. Moreover, you cannot run a Visual Process that involves other Standard or Visual Processes. Limp Mode is deactivated once the Config service becomes available.

To edit a schedule for a Visual Process

1. Click the name of the schedule you want to change or select the schedule by clicking anywhere on the row and then click .
2. On the displayed page, make the necessary changes and save the schedule.

For details about how to change the schedule settings, refer to [Adding a Schedule](#).

To delete a schedule for a Visual Process

1. Click  at the end of the row of the schedule you want to delete. A dialog box appears.
2. Click **Delete** to remove the schedule from the list.

Process Templates

Templates can be used for processes creation if the template is expected to be used in other projects for many times.

Interface, logic and instructions for templates creation and configuring using the Process Designer is the same as for processes. Refer to [Processes section](#).

Each template should have its specific name. The template name should be unique not only within templates namespace, but within the process namespace as well.

A newly created template can be saved as:

- template** — graphic elements of this templates and connections between them will be saved
- template (retain data)** — all element settings will be saved as well

Templates can be marked as favourite. The favourite templates are displayed in the Processes window on startup.

Process Logs

Process logs are used to view information about executed processes, including their start and end dates and times, duration, and the results of the execution.

User can view the Logs on the **Logs** list. The records in the log grid are editable, and the unnecessary log records can be deleted.

Currently running processes can be canceled by clicking the **Cancel** button.

To open the **Logs** list, on the Navigation Tree, double-click **Automator**, click **Visual Processes**, and select **Logs**.

The **Logs** page displays the following information:

- The log's unique ID number
- The name of the process
- The date and time when the process was started
- The date and time when the process was completed
- The duration of the process execution
- The state of the process (e.g., "Completed")
- The name of the server on which the process is run

NOTE You can filter the processes by **Log ID**, **Process Name**, **Date Started**, **Date Completed**, **State**, and **Location** by using the corresponding fields in the column headings.

The following filter settings are available:

~ - contains

== - equal

< - less

> - greater

X - Reset search value

Open a log item to view it.

The **Text** tab displays a summary of the process execution. The **Visual** tab displays the visual process of the selected process as well as the summary of the process execution. You can also view the progress of an export in the export element (shown as a percentage).

Process Events displays general information about the recently run process, including the start and end times.

Process Items allows you to filter the process information by selecting and clearing the following check boxes:

- All** — displays all file sources.
- Warning** — displays the file sources with warnings.
- Error** — displays the files sources with errors.

Click the **n Events** link below each file source to expand the relevant information, including the start and end times. The items may be sorted according to duration (ascending and descending), start time (ascending and descending), and type.

Click the **Export Log** button  to export the log to an XML file.

Click the **Back to list** button  to navigate to the Logs list.

Click the **Go to Process** button  to navigate to the Process details page (i.e., the process that was run and created in the current log).

Click the **Previous/Next** button  to navigate to the previous/next log from the Logs list.

Alerts

In the latest version of Automator, you can now define an alert for Visual Processes. Three predefined alerts can be used, and any number of additional alerts can be defined using scripting.

All existing system alerts are displayed on the **Alerts** page. To view the **Alerts** page, click **Automator > Visual Processes** and select **Alerts**.

Use this page to [add](#), [edit](#), or [delete](#) alerts.

Adding an Alert

Alerts allow users to be notified based upon certain conditions. You can, for example, be notified via email if a process fails to run. These notifications are called alerts. Some alerts are pre-defined within Automator itself, such as the process failure (SystemAlertProcessFailed) and process element failure (SystemAlertProcessElementFailed) alerts. Others can be defined by users in global, pre-export or post-export scripts.

To add an alert

1. On the **Navigation Tree**, click **Automator > Visual Processes** and select **Alerts**. The Alerts page displays.
2. On the Alerts page, click the **Add** icon  to create a new alert.
3. In the **Code** box specify the alert type.

NOTE You may type a new alert name or select one of the following predefined alert types:

- **SystemAlertProcessFailed**: Indicates that a [process](#) has failed.
- **SystemAlertNoInputToExporter**: Indicates that there was no input to the Exporter (which means, that no [input file](#) exists).
- **SystemAlertProcessElementFailed**: Indicates the failure of a project element.

If a predefined alert (for example, the SystemAlertJobFailed) has already been selected, it will no longer be listed in the Code drop-down list, as you cannot specify two predefined alerts of the same type.

4. In the **Description** field enter alert description.
5. If the alert should be immediate, check the **Rise Alert Immediate** checkbox.

6. Under **Actions**, choose which action to add to the alert.

- **Log:** Select to specify an Application Event Log notification. This can be useful if you use network management software such as BMC Patrol, IBM Tivoli, HP Openview or other software, to provide tighter integration into your systems management.
 - ◆ In the **Event level** drop-down menu, set the event as Information, Warning, or Error.
 - ◆ In the **Event ID** box, enter the event ID.
 - ◆ In the **Message** text field, type the message.

NOTE You may use the naming macros to provide a message for the log alert. To do this, click the **Insert Macro** icon  and select the naming macros from the drop-down list.

- **Email:** Select to send an email notification. Note that if the email connection is down at the time the alert is invoked, the alert will also be written to the Windows Application Event Log.
 - ◆ Specify the recipients' email addresses in the corresponding boxes: **From**, **To**, **Cc**, and **Bcc**. To select recipients from the [contacts list](#), click the **Contacts** icon . Details of the **Find Contacts** window are available in the [Email Alert Action section](#).
 - ◆ Type the message subject in the **Subject** box.

NOTE You may use the [naming macros](#) to provide a subject line for an email distribution. To do this, click the **Insert Macro** icon  and select the naming macros from the drop-down list.

- ◆ Type the message in the text box.
- ◆ Select the **Attach files** check box to attach a file for the alert.
- ◆ Select the **Zip** check box to archive the attached files. The Zip Options drop-down arrow will appear. Under Zip Options provide the following information:
 - Specify the compression type from the following: None, Medium, Maximum.
 - Select the Store directory path check box if necessary.
 - Select the Password check box to set the password.
- ◆ Select the **One file per mail** check box to attach only one file per mail.
- ◆ Select the **Attach links** check box to attach a link to file.
- ◆ Select the email priority from the following: **Low**, **Normal**, **High**.
- **RSS:** Select this to specify an RSS notification.
 - ◆ In the **Feed** box, specify the file. You can either enter the path to a file in the box or select a file by clicking the **Browse** button. The **Select Folder** dialog box will appear.
 - From the **Named Paths** list, select the appropriate named path.
 - Select the file from the list of displayed in the **Named Paths** box.
 - Click **Ok** to add the file.
 - ◆ Select the **Create if not found** checkbox in order to create the file if the specified one is not found.

- ◆ In the **Title** box, enter the item title.
- ◆ In the **Item description** box, enter item description.

NOTE You may use the [naming macros](#) to provide a title and description for an RSS distribution. To do this, click the **Insert Macro** icon  and select the naming macros from the drop-down list.

- ◆ In the **Link** box, enter the link name.

NOTE Displays the URL of the RSS item. This field is usually left blank so that Automator can automatically assign a URL (based on a [translate distribution](#)) from the input or export file that is being processed.

- ◆ Click the **Channel** title to specify the channel parameters:
 - **Title:** Enter channel title.
 - **Channel Description:** Provide channel description.
 - **Link:** Enter link name.

NOTE If you leave the **Title**, **Channel Description**, and **Link** fields blank the default values will be automatically assigned.

- **Max Items:** Specify the maximum number of items.
 - **Unlimited:** Select this check box if you want to set the unlimited number of items.
7. Click **OK** to save the action.
 8. Click **Save** to save the alert.

Editing an Alert

Once you have configured and saved an alert, you can edit it at any time.

To edit an alert

1. On the Navigation Tree, click **Automator > Visual Processes** and select **Alerts**.
2. Click the name of the alert you want to change or click anywhere on its row, and then click .

When you open a saved alert for editing, the list of actions for the alert becomes available. You can add, edit or delete actions.

For details about how to change alert settings, please refer to [Adding an Alert](#).

Deleting an Alert

You can delete any alert on the **Alerts** page.

NOTE When you delete an alert, all of its associated actions are also deleted.

To delete an alert

1. On the Navigation Tree, click **Automator > Visual Processes** and select **Alerts**.
2. Click **X** at the end of the row with the alert you want to delete. A dialog box appears.
3. Click **Delete** to remove the alert from the list.

Global Environment Scripts

Global environment scripts allow you to enter functions, classes, variables, etc., that are available for, or will execute over, all Visual Processes. They also allow the setting of global assembly imports and references, which are valid over Automator process boundaries. Global environment scripts are split into two sections:

- [Global Declarations](#): Used for variables, classes, functions, subroutines, etc., that need to be available across all Visual Processes.
- [References and Imports](#): Used for importing .NET namespaces and .NET assemblies for global use.

For global environment scripting, access the Script Editor by clicking **Automator > Visual Processes** on the **Navigation Tree** and then selecting **Scripts** from the menu.

SCRIPT EDITOR

Automator supports the use of Visual Basic.NET script in three main areas:

- [Global Environment Scripts](#): Global scripts allow users to view functions, classes, and variables, that are available or will execute over all processes. It also allows the setting of global assembly imports and references, which are valid over Automator process boundaries.
- [Project Scripts](#): Project scripts are used for project-specific scripting.
- [Process Scripts](#): Process scripts are used for process-specific scripting, enabling you to specify pre-process and post-process operations that execute before or after any operations in the process itself.

NOTE Automator users can only edit these scripts if they are granted editing privileges by the Admin.

Accessing Script Editor

As the Script Editor can be accessed from several places within Automator, its contents can vary:

- [PreExport tab](#): Specifies project-specific, pre-export declarations, e.g., to apply a pre-export script to the input file. **NOTE**: This tab is displayed only for project-specific scripting.
- [PostExport tab](#): Specifies project-specific, post-export declarations, e.g., to add steps to further process the export file(s). **NOTE**: This tab is displayed only for project-specific scripting.
- [PreProcess tab](#): Specifies process-specific, pre-process declarations. **NOTE**: This tab is displayed only for process-specific scripting.

- ❑ [PostProcess tab](#): Specifies process-specific, post-process declarations. **NOTE**: This tab is displayed only for process-specific scripting.
- ❑ [Global Declarations tab](#): Specifies Dim statements or other statements that must be global to the entire script module.
- ❑ [References and Imports tab](#): Specifies additional .NET assembly references and namespaces required by the PostExport, PreExport, and Global Declarations tabs.
- ❑ [Passwords tab](#): Specifies passwords for use in password macros.

NOTE The PreProcess, PostProcess, Global Declarations, References and Imports, and Passwords tabs are only available for Standard Processes.

You can open Help on user scripting by clicking on the **User Scripting Help** button. The button is available on the following tabs:

- ❑ PreExport tab
- ❑ PostExport tab
- ❑ PreProcess tab
- ❑ PostProcess tab
- ❑ Global Declarations tab

PreExport Script

The **PreExport** tab is part of the Script Editor. Use this tab to apply a pre-export script to the input file(s). Any script specified here will execute before any export actions take place.

The **PreExport** tab includes the following settings:

- ❑ **Script field**: Displays the current pre-export script.
- ❑ **Compile/Run**: Compiles the pre-export script and if the script is compiled without errors runs it on the specified for testing location.
- ❑ **Insert Script**: Allows you to quickly generate VB.NET script for frequently used tasks. Click to select the necessary options. For details, refer to [Inserting a Script](#) section.
- ❑ **Clear**: Removes the scripts from the script field.
- ❑ **Locations for testing**: Displays the location used for testing the script before deploying it.

PostExport Script

The **PostExport** tab is part of the Script Editor. Use this tab to add steps to further process the export file(s). Any script specified here will execute after any export actions take place and before any distributions occur.

The **PostExport** tab includes the following settings:

- ❑ **Script field**: Displays the current post-export script.
- ❑ **Compile/Run**: Compiles the script and if the script is compiled without errors runs it on the specified for testing location.
- ❑ **Insert Script**: Allows you to quickly generate VB.NET script for frequently used tasks. Click to select the necessary options. For details, refer to [Inserting a Script](#) section.
- ❑ **Clear**: Removes the scripts from the script field.

- Locations for testing:** Displays the location used for testing the script before deploying it.

PreProcess Script

The **PreProcess** tab is part of the Script Editor. Use this tab to specify any initialization or setup operations that are required before the process' jobs run.

The **PreProcess** tab includes the following settings:

- Script field:** Displays the existing pre-process declarations.
- Compile/Run:** Compiles/runs the script and if the script is compiled without errors runs it on the specified for testing location.
- Insert Script:** Allows you to quickly generate VB.NET script for frequently used tasks. Click to select the necessary options. For details, refer to [Inserting a Script](#) section.
- Clear:** Removes the scripts from the script field.
- Locations for testing:** Displays the location used for testing the script before deploying it.

PostProcess Script

The **PostProcess** tab is part of the Script Editor. Use this tab to specify any cleanup or shutdown operations to be performed after the last job of a process completes.

The **PostProcess** tab includes the following settings:

- Script field:** Displays the existing post-process declarations.
- Compile/Run:** Compiles/runs the script and if the script is compiled without errors runs it on the specified for testing location.
- Insert Script:** Allows you to quickly generate VB.NET script for frequently used tasks. Click to select the necessary options. For details, refer to [Inserting a Script](#) section.
- Clear:** Removes the scripts from the script field.
- Locations for testing:** Displays the location used for testing the script before deploying it.

Global Declarations Script for Standard Processes

The **Global Declarations** tab is part of the Script Editor. Use this tab to specify Dim statements or other statements that must be global to the entire script module.

The **Global Declarations** tab in the Script Editor for Standard Processes includes the following settings:

- Script field:** Displays the existing global declarations.
- Compile:** Compiles the global declarations script. If the script is compiled without errors, it is run on the specified testing location.
- Insert Script:** Allows you to quickly generate VB.NET script for frequently used tasks. Click to select the necessary options. For details, refer to [Inserting a Script](#) section.
- Clear:** Removes the scripts from the script field.
- User Scripting Help:** Opens the User Scripting API Help, which is a collection of script fragments that can be copied and pasted as useful starting points in the Script Editor.
- Locations for testing:** Displays the location used for testing the script before deploying it.

Global Declarations Script for Visual Processes

The **Global Declarations** tab in the Script Editor for Visual Processes includes the following settings:

- Script field:** Displays the existing global declarations.
- Insert Script:** Allows you to quickly generate VB.NET script for frequently used tasks. Click to select the necessary options. For details, refer to [Inserting a Script](#) section.
- User Scripting Help:** Opens the User Scripting API Help, which is a collection of script fragments that can be copied and pasted as useful starting points in the Script Editor.

References and Imports

The **References and Imports** tab is part of the Script Editor. Use this tab to specify additional assembly references and namespaces that are required by PostExport, PreExport, and Global Declarations sections.

The **References and Imports** tab includes the following settings:

- Namespace:** Lists the namespaces you have specified. You can add a namespace by clicking the Add button.
- Name:** Lists the names of any additional assembly references you have specified. To add a name, click the Add button.
- Assembly:** Lists the paths of any assemblies you have selected. To add a namespace, click the Add button.
- Add:** Use this button to add an assembly and name it.
- Add Named:** Displays the Add Reference dialog, with which you can name and select an assembly.
- Clear:** Removes the scripts from the script field.
- Locations for testing:** Displays the location used for testing the script before deploying it.

Passwords

This tab is part of the Script Editor. Use this tab to specify passwords for use in password macros. Specified passwords can then be utilized in pre-export or post-export scripts via the macro drop-down list, which can be accessed by clicking the Macros button on the Execute an External Command Line dialog.

The **Passwords** tab includes the following options:

- Name:** Displays the names of any specified passwords.
- Description:** Displays descriptions of any specified passwords.
- Add:** Adds a new password via the Script Password dialog.
- Edit:** Edits the selected password via the Edit Script Password dialog.
- Clear:** Removes the scripts from the script field.
- Locations for testing:** Displays the location used for testing the script before deploying it.

Inserting a Script for Standard Processes

To quickly generate VB.NET script for frequently used tasks, click the **Insert Script** button on the **Script Editor** page. The following options are available:

- Add an event to the log:** Select to add an event to the Job Log.
- Specify the text to be added to the **Job Log** in the **Event Text** field and click **Insert**.

Here you can view an example of generated script.

For example, you may wish to specify the exact nature of the export within the log, which cannot be inferred by the file name. Here is some sample code generated by the Insert Script feature to do this:

```
Log.AddEvent("Monthly General Ledger Export")
```

- ❑ **Raise an Alert:** Select to raise an alert for use with the alerts in the Automator. Specify the following settings:
 - a. In the **Name** field, select the desired alert from those listed or enter of a new alert name.

NOTE The available alerts are the user-defined alerts listed on the Automator [Alerts page](#).

- b. In the **Text** field enter the text you want displayed for the alert.
 - c. Click **Insert** to add the script.
- ❑ **Execute an external command line:** Select to execute a command line application. This function allows you to specify the command line, the parameters, including Automator [macros](#), and a timeout value for the command line. Specify the following settings:
 - a. In the **Command** field, enter the command line text.
 - b. In the **Parameters** field, specify the parameters to be passed to the program or batch file (you can also use the [naming macros](#)).
 - c. Select one of the options:
 - ◆ **No time limit:** Specifies that a time limit for the command line to complete is not necessary. **NOTE:** If the command line does not complete, Automator will wait indefinitely until it completes or is terminated.
 - ◆ **Timeout after n second(s):** Sets the delay before Automator terminates the command line.
 - d. Click **Insert** to add the script.

Here is some sample code generated by the Insert Script.

```
If Shell("""c:\test.bat""",AppWinstyle.Hide,True,10000) <>0 Then  
  'TODO: Handle command timeout here.  
End If
```

NOTE Command lines are executed by the .NET service, which for security reasons does not allow the application to show a user interface. Please ensure that any command lines or applications you run do not require a user interface and shut themselves down after completion. In addition, be sure to specify a timeout value after which Automator will kill the offending process if you do not want Automator to hold up processing altogether. If no timeout is specified, Automator will wait forever until the command line returns. If you specify a timeout that is exceeded, then you can deal with that condition in your script.

- ❑ **Iterate through a list of input or output items** (available only for project-scripting) - Select to run the script through a list of input or output items and specify whether you want to do it for data exceptions in the current project or in all projects. Specify the following settings:
 - a. **Type:** input items (iterates through a list of input items) and output items (iterates through a list of output items).

- b. **Scope:** current project (iterates through items in only the current project) and all projects (iterates through items in all projects).
- c. Click **Insert** to add the script.
- ❑ **Iterate through the list of events:** Select to iterate the script through the list of events in the job log. Simply click **Insert** to add the script.
- ❑ **Test for data exceptions** (available only for project-scripting): Select to specify whether you want to test for data exceptions in the current project or in all projects and click **Insert** to add the script.

Here is some sample code generated by the Insert Script.

In the following example, the feature has been combined with the raise alert feature to raise an alert called *Summary Exception* if any of the projects contained a summary exception as defined by the model's summary exception coloring rules.

First, the Test for Data Exceptions was chosen with an option of "All Projects" to give a script of :

```
If Log.HasExceptions() Then
  TODO: Perform desired operation if exceptions were found.
End If
```

Second, the whole line containing the TODO: was highlighted and the Insert Script To| Raise an Alert was chosen with the Alert Name of "Summary Exception" and the Alert text of "A summary exception has occurred". This produces the final script of:

```
If Log.HasExceptions() Then
  Log.AddEvent("A summary exception has occurred", "Summary Exception")
End If
```

Now you can define the [actions for this alert](#) on the Alerts page of the Automator.

NOTE: You must have already defined summary exception coloring rules in your Monarch model in order for this feature to work.

- ❑ **Set the value of a runtime field:** Select to specify a value for a runtime field. Specify the following settings:
 - a. In the **Field Name** field, enter the name of the runtime field.
 - b. In the **Field Value** field, specify the desired value for the runtime field.
 - c. Select one of the following:
 - **Set this value unconditionally:** Sets the value unconditionally, resulting in a simple call to `SetRuntimeFieldValue`.
 - **Set this value only if a value for this field has not already been set:** Sets the value conditionally, resulting in a call to `SetRuntimeFieldValue` that is contingent on a return of `False` from `HasRuntimeFieldValue`.
 - d. Click **Insert** to add the script.
- ❑ **Export to MapR Hive table:** Select to export data to MapR Hive table. Specify the following settings:
 - a. Specify the target server information: **Host Name or IP Address**, **Cluster Name** and path to the **HDFS Directory** (if necessary).
 - b. Specify the export information: **Project ID** and **Export Name**.
 - c. To overwrite data in the existing table, select the **Overwrite Table** check box.
 - d. Click **Insert** to add the script.
- ❑ **Export to HD Insight Hive table:** Select to export data to HD Insight Hive table. Specify the following settings:
 - a. Specify the target server information: **Host Name or IP Address** and path to the **HDFS Directory** (if necessary).

- b. Specify the export information: **Project ID** and **Export Name**.
- c. To overwrite data in the existing table, select the **Overwrite Table** check box.
- d. Click **Insert** to add the script.
- Copy to HDFS MapR:** Select to copy data to HDFS MapR.
 - a. Specify the target server information: Host Name or IP Address, Cluster Name, Target Directory Path and Target File Name.
 - b. Specify the export information: **Project ID** and **Export Name**.
 - c. Click **Insert** to add the script.
- Copy to HDFS HD Insight:** Select to copy data to HDFS HD Insight. Specify the following settings:
 - a. Specify the target server information: Host **Name** or **IP Address**, **Target Directory Path** and **Target File Name**.
 - b. Specify the export information: **Project ID** and **Export Name**.
 - c. Click **Insert** to add the script.
- Insert:** Inserts the current pre-export script into the script field.
- Cancel:** Cancels any changes you have made.

Inserting a Script for Visual Processes

To quickly generate VB.NET script for frequently used tasks, click the **Insert Script** button. The following options are available:

- Add an event to the log:** Select to add an event to the log. Enter the event text in the field provided.
- Raise an alert:** Select to prompt an alert to the process.
 - a. In the **Code** field, enter the name of the alert code.
 - b. In the **Message** field, provide an alert message.
- Execute an external command line:** Select to execute a command line application. This function allows you to specify the command line, the parameters, including Automator [macros](#), and a timeout value for the command line. Specify the following settings:
 - a. In the **Command** field, enter the command line text.
 - b. In the **Parameters** field, specify the parameters to be passed to the program or batch file (you can also use the [naming macros](#)).
 - c. Select one of the options:
 - ◆ **No time limit:** Specifies that a time limit for the command line to complete is not necessary.

NOTE If the command line does not complete, Automator will wait indefinitely until it completes or is terminated.

- ◆ **Timeout after n second(s):** Sets the delay before Automator terminates the command line.
- d. Click **Insert** to add the script.

Here is some sample code generated by the Insert Script.

```
If Shell("""c:\test.bat""",AppWinstyle.Hide,True,10000) <>0 Then
'TODO: Handle command timeout here.
```

End If

NOTE Command lines are executed by the .NET service, which for security reasons does not allow the application to show a user interface. Please ensure that any command lines or applications you run do not require a user interface and shut themselves down after completion. In addition, be sure to specify a timeout value after which Automator will kill the offending process if you do not want Automator to hold up processing altogether. If no timeout is specified, Automator will wait forever until the command line returns. If you specify a timeout that is exceeded, then you can deal with that condition in your script.

- Iterate through a list of input or output items** — Select to run the script through a list of input or output items and specify whether you want to do it for data exceptions. Specify the following settings:
 - a. Iterate through input items.
 - b. Iterate through output items.
 - c. **Export Name** — type the export element name.
- Set the value of a runtime field:** Select to specify a value for a runtime field. Specify the following settings:
 - a. In the **Field Name** field, enter the name of the runtime field.
 - b. In the **Field Value** field, specify the desired value for the runtime field.
 - c. Select one of the following:
 - ◆ **Set this value unconditionally:** Sets the value unconditionally, resulting in a simple call to `SetRuntimeFieldValue`.
 - ◆ **Set this value only if a value for this field has not already been set:** Sets the value conditionally, resulting in a call to `SetRuntimeFieldValue` that is contingent on a return of `False` from `HasRuntimeFieldValue`.
 - d. Click **Insert** to add the script.
- Insert Script:** Inserts the current script into the script field.
- Cancel:** Cancels any changes you have made.

Testing a Script

You can test your PreProcess/PostProcess or PreExport/PostExport script before deploying it. This allows you to run the script based on a Job Log file you specify.

After entering a script in the Script Editor's PreExport or PostExport tab, or the PreProcess or PostProcess tab, click the **Compile/Run** button and, if your script is compiled without any errors, click the **OK** button in the Run Script dialog box that appears.

The **Run Script** dialog box includes the following settings:

Entry Point

- PreExport** (Log as JobLog, Project ID as Integer): Specifies that you want to run the PreExport function.
- PostExport** (Log as JobLog, Project ID as Integer, ExportCompleted as Boolean): Specifies that you want to run the PostExport function.
- PreProcess** (Log as JobLog): Specifies that you want to run the PreProcess function.
- PostProcess** (Log as JobLog): Specifies that you want to run the PostProcess function.

Parameters

- Job ID:** Displays the Job ID of the job log to which the script will be applied. **NOTE:** To select a job ID, click its name in the table or select from the Select Job ID drop-down list.
- Project ID:** Specifies the Project ID to pass to the PreExport or PostExport Function. **NOTE:** This only needs to be changed if you are using the Multiple Projects per Job option in the [Process settings](#). Otherwise leave the value at 1.
- Export Completed:** Allows you to specify that the ExportCompleted variable is forced to true, if checked, and false, if unchecked, so any code checking the contents of the variable can be fully tested, even if an export fails.
- Run:** Runs the selected function.
- Show Log:** Displays the contents of the selected log file. **NOTE:** After the function runs, you may wish to click this button to view the log file to make sure that the function modified the log file in the desired way.
- Cancel** - Closes the dialog box and returns to the Script Editor page.

CONNECTION DEFINITIONS

The **Connection Definitions** tab is part of the Project Editor. Use this tab to add predefined/saved connection definitions that can be reused in multiple projects.

Important: Connection definition parameters are taken from the server at a project creation time. Any changes in initial connection strings on source machines will not be tracked.

To add connection definition

1. On the Navigation Tree, click Automator and select Connection Definitions.
2. On the displayed page, click the **Add** icon . The **Connection Definition** page appears.
3. Enter the connection definition name in the **Name** field.
4. Enter connection definition description in the **Description** field.
5. Enter connection string in the **Connection String** field:
 - Manually and then proceed to step 6, or
 - Click the Browse button to open Connection Type Wizard window:
 - a. Select provider type from the following choices: OleDb for Native MsSql Provider, OleDb Provider or Odbc Provider
 - b. Provide parameters according to provider type chosen
 - c. Click the **Test Connection** button to test whether or not the data link was successful (i.e., to test whether you can connect to the specified OLE DB or ODBC provider). If the connection is successful, a Test Connection Succeeded message is displayed.
 - d. To use connection string, click **OK**.
6. Select **Include Password** check box if your connection requires password.
7. Enter matching passwords in the **Password** and **Confirm Password** fields.
8. Click the **Test Connection** button to make sure that your DB connection is successful.
9. To save connection definition, click **Save**.

To duplicate connection definition

1. On the Navigation Tree, click **Automator** and select **Connection definitions**.
2. On the displayed page, select the name of the connection definition that you want to duplicate.

3. Click the **Duplicate** icon . The **Connection Definition** page appears.
4. Enter connection definition name in the **Name** field. The name must not coincide with the original definition name. All other settings of this definition will be duplicated.
5. To save connection definition, click **Save**.

To manage security rights for a connection definition

1. On the **Navigation Tree**, click **Automator** and select **Connection Definitions**.
2. On the displayed page, select the connection definition whose security settings you want to modify.
3. On the **Connection Definition** page, click the **Rights** tab.
4. If you want to make the connection definition available to all users, click the **Global** checkbox.
5. If you want to make the connection definition available to specific user groups, roles, or users, click the corresponding tabs to select the user groups, roles, and users you want to grant access to the connection definition.
6. Click **Save** to save the settings, **Clear** to reject all changes made, and **Cancel** to stop the operation and return to the **Connection Definition** page.

USING THE NAMING MACROS

Automator offers a number of naming macros that can be used to name files and tables that are being exported, moved, copied and so on. They can also be used to add information when performing distributions or even when using scripting. The macros can be used individually or in combination with one another.

NOTE Some of the naming macros may not be valid under certain conditions (for example, some macros are only available when using scripts), so until you are well acquainted with the product, use the **Insert Macro** icon  to ensure that you select macros that are valid for the situation.

The following naming macros are available:

MACRO	SYMBOL	RESULT
Year (4 digit)	&[yyyy]	4-digit year (for example, 2007)
Year (2 digit)	&[yy]	2-digit year (for example, 07)
Month	&[mm]	Month
Day	&[dd]	Adds the day to the file name as a two digit numeral, for example, 09 for the ninth Day
Hour	&[hh]	Hour
Minute	&[mi]	Minutes

MACRO	SYMBOL	RESULT
Second	&[ss]	Seconds
Quarter	&[qtr]	1-digit quarter of the calendar year: 1 for dates in January: March, 2 for dates in April: June, 3 for dates in July: September, 4 for dates in October: December.
Day of Year	&[dayofyear]	3-digit day of the year in range 001-366.
Previous Day	&[dd.-1d]	2-digit value in the range of 01-31.
Previous Month	&[mm.-1m]	2-digit value in the range of 01-12.
Previous Year	&[yyyy.-1y]	4-digit year (referring to previous year).
ISO Year	&[isoyear]	4-digit year corresponding to the ISO-8601 week in which the date falls.
ISO Week	&[isoweek]	2-digit week of the year (range 01-53) as defined in ISO-8601. Note that it's possible for a date in late December to actually fall in week 01 of the following year. Likewise, it's possible for a date in early January to fall in the last week (52 or 53) of the previous year.
ISO Day of Week	&[isoday]	1-digit day of the week as defined in ISO-8601, namely, Monday=1, Tuesday=2, Wednesday=3, ... Sunday=7.
Summary Name	&[summary]	Name of current summary. Empty if no summary is current.
Filter Name	&[filter]	Name of current filter. Empty if no filter is current.
Measure Title	&[measure]	Name of current measure. Empty if no measure is current.
Key Value	&[keyvalue]	Value of a key field.
First Value of Field	&[firstvalue.value name]	Name of the first value in the selected field. See the Select a Field Name dialog for more information.
Input File Location	&[input]	Full path and filename of an input file. To specify multiple input files use the convention &[input.1] &[input.2] etc. If there are more &[input]s than input files, the extra ones are replaced by empty strings. When importing from a database there are no input files, so all &[input]s are replaced by empty strings.
Input File Name	&[inputname]	Filename of an input file. To specify multiple input files use the convention &[inputname.1] &[inputname.2] etc. If there are more &[inputname]s than input files, the extra ones are replaced by empty strings. When

MACRO	SYMBOL	RESULT
		importing from a database there are no input files, so all <code>&[inputname]</code> s are replaced by empty strings.
Output File Location	<code>&[output]</code>	Full path and filename of an export file. To specify multiple output files use the convention <code>&[output.1]</code> <code>&[output.2]</code> etc. If there are more <code>&[output]</code> s than export files, the extra ones are replaced by empty strings.
Distribution Source file path	<code>&[source]</code>	Full path and filename of a file that is being distributed via a distribution action. If multiple files exist, this macro will expand all names and concatenate them using a semicolon.
Distribution Source file path (Advanced)	<code>&[source.option]</code>	The option specifies which part of the path to return, i.e., <code>dir</code> for directory, <code>file</code> for filename with extension, <code>name</code> for filename alone, <code>ext</code> for extension.
Job Number	<code>&[jobid]</code>	Job number
Job Log	<code>&[joblog]</code>	Full path and filename of the Job Log
Process Name	<code>&[process]</code>	Process name
Date Format	<code>&[date.<date increment>.<output format>]</code>	Subtracted date; <date increment> could be <code>-1d</code> , <code>+1d</code> , <code>-1m</code> , <code>+1m</code> , <code>-1y</code> , <code>+1y</code> , and <output format> could be <code>yyyyMMdd</code> , <code>yyyy-MM-dd</code> , etc. The output format is case-sensitive. For a full list of supported format specifiers, click here .
System-Wide Counter	<code>&[counter]</code>	System counter. Automatically increments with each use — counts across all jobs in all processes.
Item Addressing (Advanced)	<code>&[item.xpath]</code>	A specific element or attribute of a job log input or output item. The XPath following the macro name is an XPath expression describing exactly which node is to be returned. Note that these macros are only available for scripting and distributions. In most cases they will be used in scripting to assist in navigating through the job log. They cannot be used for file or table naming.
Item File Path	<code>&[source]</code>	Full path of an item.
Item File Directory	<code>&[source.dir]</code>	Full path of item directory.
Item File Name	<code>&[source.file]</code>	File name of an item.
Item File Name (no ext.)	<code>&[source.name]</code>	File name of an item with no extension.

MACRO	SYMBOL	RESULT
Item File Extension	&[source.ext]	Extension of item file.
Item URL	&[source.uri]	URL of an item.
Item Title	&[item.title]	Title of an item.
Item Description	&[item.description]	Description of an item.
Item Addressing Examples	&[item.title]	Returns the auto-generated item title
	&[item.description]	Returns the auto generated item description
	&[item.sort]	Returns the sort, if specified
	&[item.filter]	Returns the filter, if specified
	&[item.summary]	Returns the summary, if one exists
	&[item.totals/@record _count	Returns the record count

The naming macros are available in the following locations:

- [Job Disposition Settings](#): Use the naming macros here to name job logs that have been moved.
- [Copy Distribution Action](#): Use the naming macros here to name copies of files generated by a distribution.
- [Move Distribution Action](#): Use the naming macros here to name files that have been moved via a distribution.
- Email Action: Use the naming macros here to provide a subject line for an email [distribution](#) or [alert](#).
- RSS action: Use the naming macros here to provide a title and description for an RSS [distribution](#) or [alert](#).
- [Default Job Log Disposition](#): Use the naming macros here to assign new names to the job logs when moving them.

NOTE The naming macros can also be used when scripting, for example, when using the Item.ExpandMacros method. See the UserScripting Help file for additional information. To use the naming macros in email body fields and NT Event Log Message fields, right-click and choose Insert.

Utilizing naming macros

1. When viewing a field for which you can select a naming macro, click the adjacent **Insert Macro** icon  to view the naming macro drop-down list.
2. Select the desired naming macro from the list.

Syntax checking of macros

In the Export Wizard and the Portable Report Settings dialog, if the file name contains a macro (or macros), Automator performs a series of tests to be sure that the macros are well-formed. For example, the following conditions are detected and reported as errors:

- ❑ Unknown macro names, for example, `&[fred]`. Such macros will not expand at runtime.
- ❑ Parameterized macros having ill-formed parameters, for example, `&[yyyy.+1q]`. Such macros will expand to empty at runtime.
- ❑ Macro names that are not appropriate in the context of the export in question. For instance, `&[filter]` makes no sense when publishing a PRF file.
- ❑ Ill-formed macros. For example, a macro prefix (`&`) without its matching suffix (`[]`), or a macro embedded within a macro for example, `&[fil&[dd]ter]`.

OFFSET EXPRESSIONS IN MACROS

The date/time macros (yyyy, mm, etc.) have been extended by defining "offset expressions" that may be tacked onto the macro name to compute an offset from today's date.

NOTE Offset expressions are available for file naming only; they are not available for distributions, etc.

The syntax is as follows:

```
&[macro{.sign{count}units}]
```

- ❑ The braces (`{ }`) enclose optional portions of the syntax.
- ❑ *macro* is a macro name (e.g., yyyy).
- ❑ *sign* is + (plus) to add to the current date or - (minus) to subtract from the current date.
- ❑ *count* is an optional integer count.
- ❑ *units* specifies the units to be added or subtracted: d for days, m for months or y for years.

NOTE The offset expression, if present, must be separated from the macro name by a period (.).

Syntax for Offset Expressions

The following table provides examples of syntax for offset expressions:

EXAMPLE	MEANING
<code>&[dd.+1d]</code>	The day portion of tomorrow's date.
<code>&[dd.+d]</code>	Synonymous with <code>&[dd.+1d]</code> . The count defaults to 1.
<code>&[mm.+1d]</code>	The month portion of tomorrow's date.
<code>&[yyyy.+1d]</code>	The 4-digit year portion of tomorrow's date.
<code>&[dd.-1d]</code>	The day portion of yesterday's date

EXAMPLE	MEANING
&[mm.+7d]	The month portion of the date one week from today
&[mm.+1m]	The month portion of the date one month from today.
&[yyyy.+1y]	The year portion of the date one year from today.
&[yyyy.+2m+15d]	The year portion of the date 2 months plus 15 days from today. More than one offset expression may be concatenated as shown to affect sums of different units.

NOTE The offset expression may be used with any of the date/time macros, including the time macros (hh, mi, and ss), although they have no effect on the latter, since **&[hh.+1d]** would give the same result as **&[hh]**. Likewise, one may make a macro like **&[mm.+4y]**, but it is equivalent to **&[mm]** because adding or subtracting an integral number of years to a date can never affect the month.

WHEN A JOB FAILS TO RUN

A [job](#) may fail to run for a number of reasons, including the following:

- ❑ **The Automator Service is not running:** Check the Server Status page and make sure that the service is running.
- ❑ **No input files are in the input folder:** When you create a process, you specify a folder where the input file you want to use is located (or will be located). If the File Existence Criteria you specified for the project is that at least one input file must exist or that all files must exist, the job will not run until an input file of the specified type appears in the input folder.
- ❑ **Associated report file is no longer available:** If the report file associated with the job's project is no longer present, the job will be halted.
- ❑ **Project file is no longer available:** If you attempt to run a process for which the project file no longer exists or has been moved, the associated job will fail to run.
- ❑ **Process prerequisites have not been met:** If you specified a process prerequisite, the associated job will not run until that prerequisite has been met, regardless of when you scheduled the process to run.

NOTE For each job that is spawned a log file is created. If a job fails to run, you should examine the job's log, as it should provide information as to why the job failed.

RUNTIME FIELDS

Runtime fields are a special type of calculated field whose value can be easily set at runtime, thereby providing a way to parameterize a model in some fashion. When Monarch is running in interactive mode it prompts the user via a

dialog box for the values of any runtime fields at the time a model is loaded. Runtime fields may also be used with Monarch in "lights-out" mode by specifying the desired values on the command line or via an automation method.

With Automator, runtime field values may be specified via script, or a manifest file when using the Web Service or API. Regardless of the source, the runtime field values are written to a special section of the job log where they are then read at the time the project is loaded.

Specifying Runtime Values via a Manifest

To specify values for runtime fields using a manifest, construct the manifest XML as usual, but add a <runtime fields> section structured as in the following example:

```
<manifest process="ClassicWithRuntime">
  <inputs />
  <runtime_fields>
    <runtime_field name="Interest Rate">1.75%</runtime_field>
    <runtime_field name="SelectedComposer">Mozart</runtime_field>
  </runtime_fields>
</manifest>
```

The name attribute of each runtime_field tag is the field name exactly as it appears in the model, and the text is the desired value. One may specify as many runtime field values as desired. The values specified in a manifest apply across all projects/models in the specified process because Automator automatically copies these values to the job logs of any and all jobs spawned by the process instance. Once a suitable manifest has been constructed it may be passed into Automator in a call to StartProcessUsingManifest, which method already exists both in Automator's .Net API and in its Web Service.

Specifying Runtime Values via Script

One may also specify runtime values in PreExport, PreProcess, or PostExport script. In the most likely scenario, one would set the desired values in the PreExport script of the project to which they apply. However, in a multiple-project-per-job scenario it might be convenient to set the values in the PreProcess script, in which case the values would apply across all projects/models in the process. In rare cases, it may even be useful to set values in PostExport script, in which case the values would apply to any projects/models that are "downstream" of that project in the same job.

The following methods are included in the JobLog class to support getting/setting of runtime field values:

METHOD	DESCRIPTION
SetRuntimeFieldValue(string strFieldName, string strFieldValue)	Sets the value of a specified runtime field, superseding the old value, if any, which may have been set for that field.
HasRuntimeFieldValue(string strFieldName)	Returns true if a value has already been set for the specified runtime field.
GetRuntimeFieldValue(string strFieldName)	Returns the value of the specified runtime field as a string. Returns an empty string if no value has yet been set for the field.
SetRuntimeFieldValues(string[] FieldNames, string[] FieldValues)	Sets the values for a whole set of runtime fields in one call. This is used internally by Automator but might also be useful to a sophisticated script programmer.

METHOD	DESCRIPTION
GetRuntimeFieldValues(out string[] FieldNames, out string[] FieldValues)	Gets the runtime field names and their corresponding values for all fields whose values have already been set.

To make the setting of runtime field values easily accessible, Automator includes a **Set the value of a runtime field...** option in the Script Editor's **Insert script to...** menu. If you select this option, the **Set the Value of a Runtime Field** dialog appears, in which you may enter the name of the field and its desired value. In addition, you may choose whether the value is set unconditionally (resulting in a simple call to SetRuntimeFieldValue) or conditionally (resulting in a call to SetRuntimeFieldValue which is contingent on a return of false from HasRuntimeFieldValue).

NOTE The **Set the Value of a Runtime Field** dialog makes no attempt to enumerate the possible runtime fields, nor does it check that the given field name actually corresponds to a runtime field in the model(s) in question.

Runtime Fields in the Job Log

Whenever runtime field values are used in Automator, they are recorded in the job log. This serves two purposes:

- It is the mechanism by which the values are communicated to Exporter
- It provides documentation for what values were actually in effect during the export(s)

The runtime field values appear in the job log in a <runtime fields> section structured exactly the same as the corresponding section of a manifest (see earlier in this section). The job log's <runtime_fields> section records the last value set for each runtime field. In addition, whenever a runtime field value is set, either indirectly via a manifest or directly in script, an event is automatically added to the <events> section of the log. This event documents the field name and the value set for it at that point in the execution, thereby making it possible to accurately determine at a later time what value a particular runtime field had at the time of a particular export.

PROJECT INPUTS VIA MANIFEST

To specify values for Text Inputs using a manifest, construct the manifest XML as in the following example:

```
<manifest process="Process">
  <inputs>
    <item name="Input">
      <uri>file:Input path</uri>
    </item>
  </inputs>
</manifest>
```

NOTE JSON schema may also be used for manifests. An example is shown below.

```
{
  "process": "Process",
  "inputs": [
```

```
{
  "name": "Input",
  "uri": "Input path"
}
```

To specify values for Database Inputs using a manifest, construct the manifest XML as in the following example:

```
<manifest process="Process">
  <inputs>
    <item name="Input">
      <db_import excel_column_names_row="true or false" replace_nulls="true or false">
        <data_source>data source</data_source>
        <table_name>table name</table_name>
        <password>password</password>
        <apply_filter>filters</apply_filter>
      </db_import>
    </item>
  </inputs>
</manifest>
```

NOTE

JSON schema may also be used for this purpose. An example is shown below.

```
{
  "process": "Process",
  "inputs": [
    {
      "name": "Input",
      "db_import": {
        "data_source": "data source",
        "table_name": "table name",
        "password": "password",
        "replace_nulls": "true or false",
        "excel_column_names_row": "true or false",
        "apply_filter": "[column name]=value"
      }
    }
  ]
}
```

PROJECT OUTPUTS VIA MANIFEST

In Standard Processes, projects can be added, removed, or sorted via manifest. The manifest XML for this purpose should be prepared as in the following format:

```
<manifest process="Process">
```

```

<projects>
  <add>
    <project>Path to Project 1</project>
    <project>Path to Project 2</project>
  </add>
  <remove>
    <project>Path to Project 3</project>
    <project>Path to Project 4</project>
  </remove>
  <order>
    <project order="1">Path to Project 5</project>
    <project order="2">Path to Project 2</project>
    <project order="3">Path to Project 3</project>
  </order>
</projects>
</manifest>

```

NOTE

JSON schema may also be used for manifests. An example is shown below.

```

{
  "process": "Process",

  "projects": {
    "add":["Path to Project 1", "Path to Project 2"],
    "remove":["Path to Project 3", "Path to Project 4"],
    "order":[{"order:1, value: "Path to Project 5"}, {order:2,
value: "Path to Project 2"}, {order:3, value: "Path to
Project 3"}]
  }
}

```

To override the output destination of a report export, construct the manifest XML as in the following example:

```

<manifest process="Process">
  <outputs>
    <item project="path to project" name="ReportExport">
      <data_source>Report export path</data_source>
    </item>
  </outputs>
</manifest>

```

NOTE

JSON schema may also be used for this purpose. An example is shown below.

```

{
  "process": "Process",

  "outputs": [
    {
      "name": "ReportExport",
      "project": "path to project",
      "data_source": "Report export path"
    }
  ]
}

```

To override the output destination of a table export, construct the manifest XML as in the following example:

```

<manifest process="Process">
  <outputs>
    <item project="path to project" name="TableExport">
      <data_source>Table export path</data_source>
      <table_name>Table name</table_name>
    </item>
    <item project="path to project" name="TableExport2">
      <data_source>Provider=SQLNCLI11;Data Source=localhost;User
ID=sa;Initial Catalog=Export</data_source>
      <table_name>Table name</table_name>
      <password>Password</password>
    </item>
  </outputs>
</manifest>

```

NOTE

JSON schema may also be used for this purpose. An example is shown below.

```

{
  "process": "Process",

  "outputs": [
    {
      "name": "TableExport",
      "project": "path to project",
      "data_source": "Table export path",
      "table_name": "Table name"
    },
    {
      "name": "TableExport2",
      "project": "path to project",
      "data_source": "Provider=SQLNCLI11;Data
Source=localhost;User ID=sa;Initial Catalog=Export",
      "table_name": "Table name",
      "password": "Password"
    }
  ]
}

```

To override the output destination of a summary export, construct the manifest XML as in the following example:

```

<manifest process="Process">
  <outputs>
    <item project="path to project" name="SummaryExport">
      <data_source>Summary export path</data_source>
      <table_name>Table name</table_name>
    </item>
    <item project="path to project" name="SummaryExport2">
      <data_source>Provider=SQLNCLI11;Data Source=localhost;User
ID=sa;Initial Catalog=Export</data_source>
      <table_name>Table name</table_name>
      <password>Password</password>
    </item>
  </outputs>
</manifest>

```

NOTE

JSON schema may also be used for this purpose. An example is shown below.

```
{
  "process": "Process",

  "outputs": [
    {
      "name": "SummaryExport",
      "project": "path to project",
      "data_source": "Summary export path",
      "table_name": "Table name"
    },
    {
      "name": "SummaryExport2",
      "project": "path to project",
      "data source": "Provider=SQLNCLI11;Data
Source=localhost;User ID=sa;Initial Catalog=Export",
      "table_name": "Table name",
      "password": "Password"
    }
  ]
}
```

STANDARD PROCESSES WITH MANIFEST MONITORING

In manifest monitoring in Standard Processes, a manifest XML file should be prepared in the following format:

```
<manifest process="ProcessName">
  <inputs>
    <item name="InputName">
      <uri>file:Input path</uri>
    </item>
  </inputs>
</manifest>
```

NOTE

[XSD](#) can be used to prepare the manifest XML for Standard Processes.

An example of an XML file is shown below.

```
<manifest process="m">
  <inputs>
    <item name="FirstReport">
      <uri>file:C:\Test Scenarios\Test Scenario ManifestFile\Classic1.prn</uri>
    </item>
  </inputs>
</manifest>
```

```

</item>
<item name="SecondReport">
  <uri>file:C:\Test Scenarios\Test Scenario ManifestFile\Classic6.prn</uri>
</item>
<item name="ThirdReport">
  <uri>file:C:\Test Scenarios\Test Scenario ManifestFile\Classic21.prn</uri>
</item>
</inputs>
</manifest>

```

NOTE

The process name in the XML file ("m" in the example above) should be the same as the name of the Standard Process created.

Other than XML, the following JSON structure may also be used:

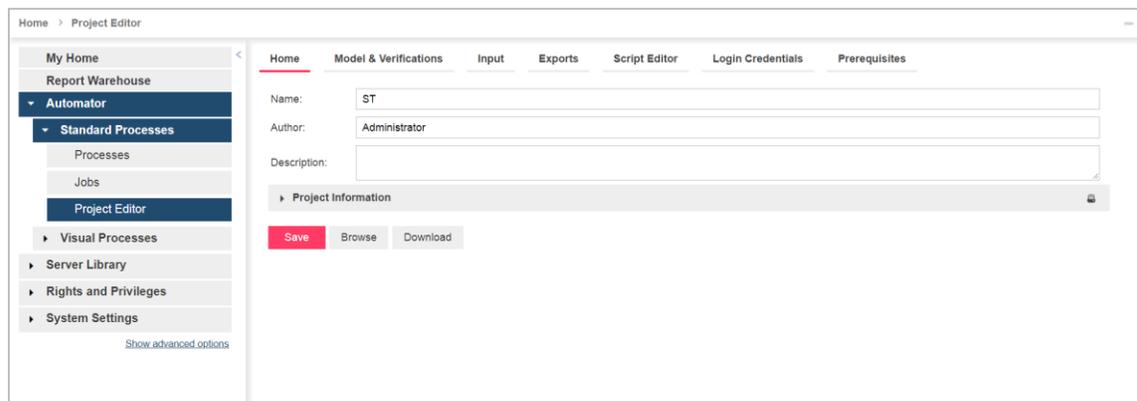
```

{
  "process": "ProcessName",
  "inputs": [
    {
      "name": "InputName",
      "uri": "Input path"
    }
  ]
}

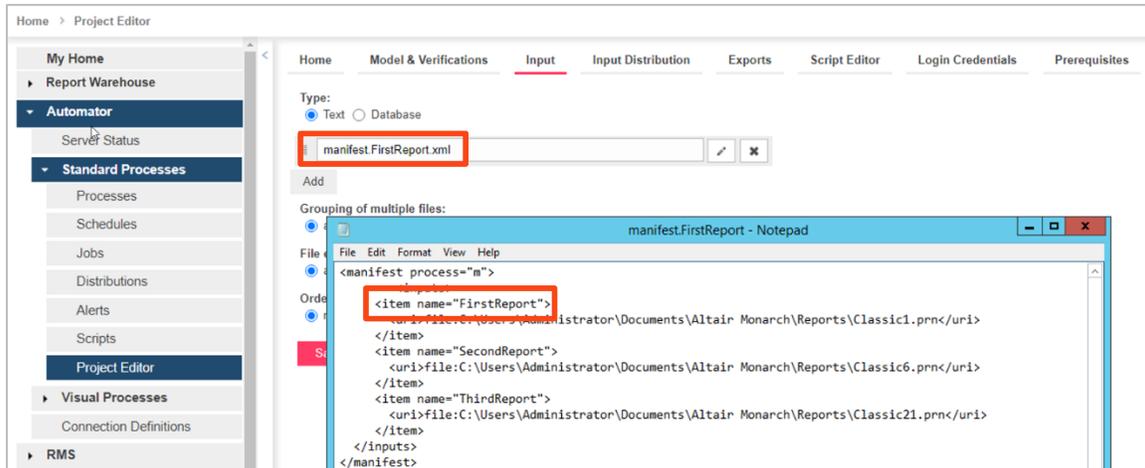
```

To proceed with manifest monitoring

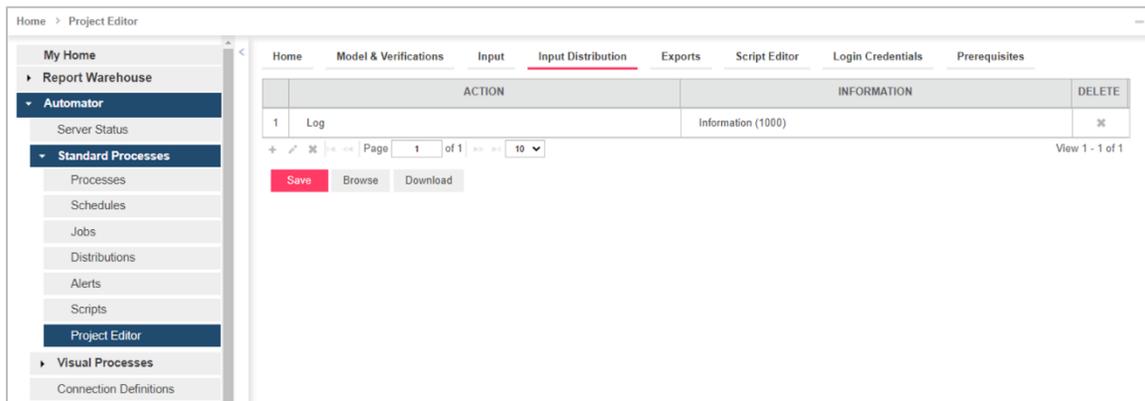
1. Open **Project Editor**. Create a new project, and assign it with any name.



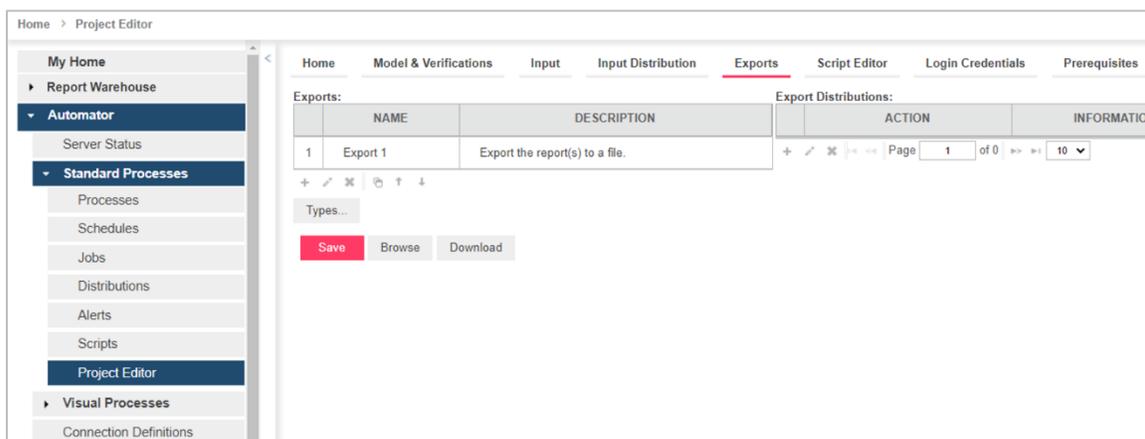
2. Open the **Model & Verifications** tab, and set the model for the new project. See [Specifying Model Settings for a Project File](#).
3. Open the **Input** tab to add an input file. If you would like to use an xml file as the input file, rename the file as "manifest.xxx," where "xxx" is the item name from the xml file.



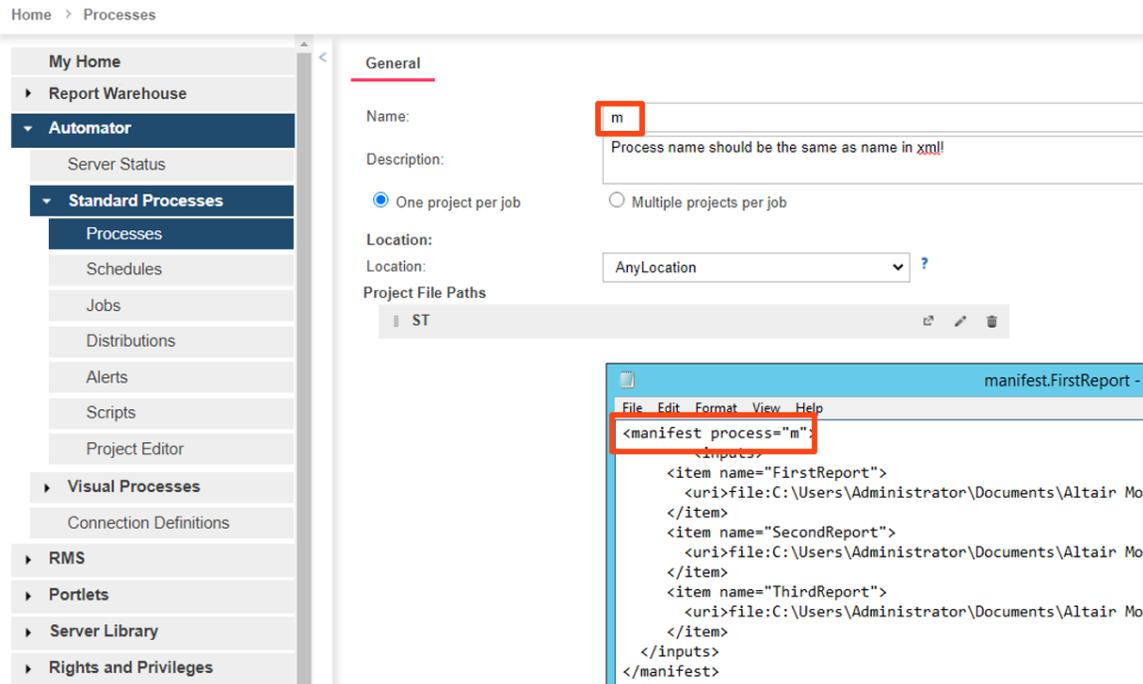
4. Add any Input Distribution.



5. Add any Export. Tap on **Commit project**, and then click **Save**.



6. Under **Standard Processes** in the Navigation Tree, click **Processes**, and create a new **Standard Process** (see [Adding a Process](#)). Add the project you created in Step 1.



7. Click **Save**.

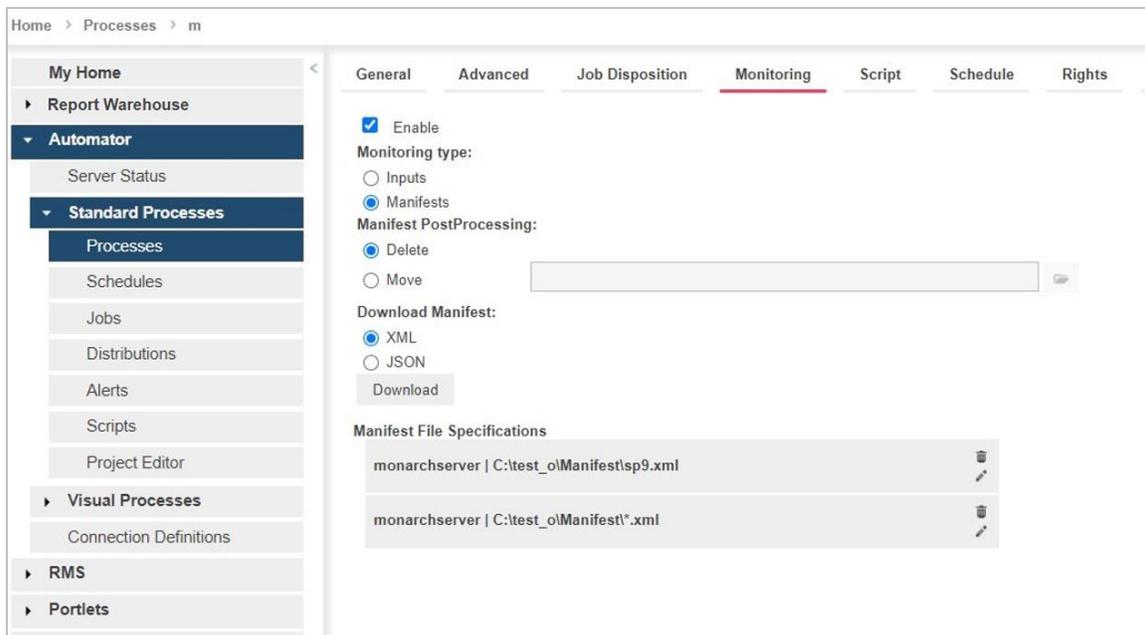
NOTE The process name in the XML file ("m" in the example above) should be the same as the name of the Standard Process created.

8. Open the process you created in Step 6.

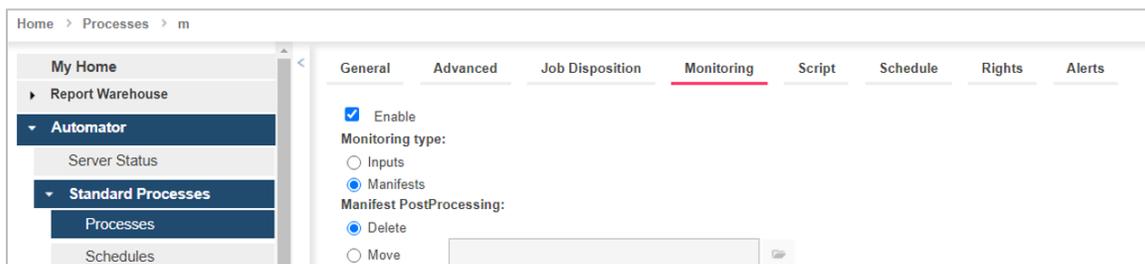
In the **Monitoring** tab, click the **Enable** check box, and select **Manifests** under **Monitoring Type**.

9. Add the manifest xml file you prepared (as described in the introduction of this section).

NOTE You may use a wildcard (* or ?) to select any file (e.g., *.xml).



10. Select one of two options in Manifest PostProcessing.
- Delete (selected by default): Removes the XML file after processing
 - Move (and set the folder path): Moves the XML file after processing



11. **Save** the process.
12. Proceed to the **Jobs** page under **Standard Processes** to view the job log or re-run the job.

NOTE

The **manifest xml** file may only be run again in the following three cases:

- When changes are made to the xml file
- When a new manifest monitoring file is added to a process
- When a new manifest monitoring file is added to the folder after using a wildcard (* or ?) in the name of the manifest file

Under these cases, the manifest XML file will automatically run again.

JOB ID	PROCESS NAME	LOG	STATUS	LOCATION	REGISTERED	LAST PROCESSED	DURAT
110	m		Job completed without errors.	monserv13-5	08/16/2020 7:31:	08/16/2020 7:31:	00:00
109	February Sales		Process prerequisites failed, but no	monserv13-5	08/10/2020 11:28	08/10/2020 11:28	00:00
108	January Sales		Distribution completed with errors.	monserv13-5	08/10/2020 10:57	08/10/2020 10:57	00:00
107	February Sales		Job completed without errors.	monserv13-5	08/10/2020 10:55	08/10/2020 10:55	00:00
106	January Sales		Job completed without errors.	monserv13-5	08/09/2020 11:39	08/09/2020 11:39	00:00
105	January Sales		Job completed without errors.	monserv13-5	08/09/2020 8:02:	08/09/2020 8:02:	00:00
104	Test 4		Job completed without errors.	monserv13-5	08/09/2017 12:52	08/09/2017 12:52	00:00
103	Test 3		Process prerequisites failed, but no	monserv13-5	08/09/2017 12:48	08/09/2017 12:48	00:00
102	Test 2		Job completed without errors.	monserv13-5	08/09/2017 12:36	08/09/2017 12:36	00:00
101	Test 1		Process prerequisites failed, but no	monserv13-5	08/09/2017 12:33	08/09/2017 12:33	00:00

VISUAL PROCESSES WITH MANIFEST MONITORING

In manifest monitoring in Visual Processes, a manifest XML file should be prepared in the following format:

```
<?xml version="1.0" encoding="utf-8"?>
<ProcessStartParams ProcessName="Test name">
  <ProcessItems>
    <ProcessItem>
      <ItemType>InputFile</ItemType>
      <FileInput>
        <Name>File Input</Name>
        <FileLocation>file:Input path</FileLocation>
        <PdfPassword/>
      </FileInput>
    </ProcessItem>
  </ProcessItems>
</ProcessStartParams>
```

NOTE [XSD](#) can be used to prepare the manifest XML for Visual Processes.

An example of an XML file is shown below.

```
<?xml version="1.0" encoding="utf-8"?>
```

```

<ProcessStartParams ProcessName="Test name">
  <ProcessItems>
    <ProcessItem>
      <ItemType>InputFile</ItemType>
      <FileInput>
        <Name>File Input</Name>
        <FileLocation>\\ServerName.com\c$\test_o\Classic1.prn</FileLocation>
        <PdfPassword/>
      </FileInput>
    </ProcessItem>
  </ProcessItems>
</ProcessStartParams>

```

NOTE

- The process name in the XML file ("Test name" in the example above) should be the same as the name of the Visual Process created.
- The following JSON structure may also be used:

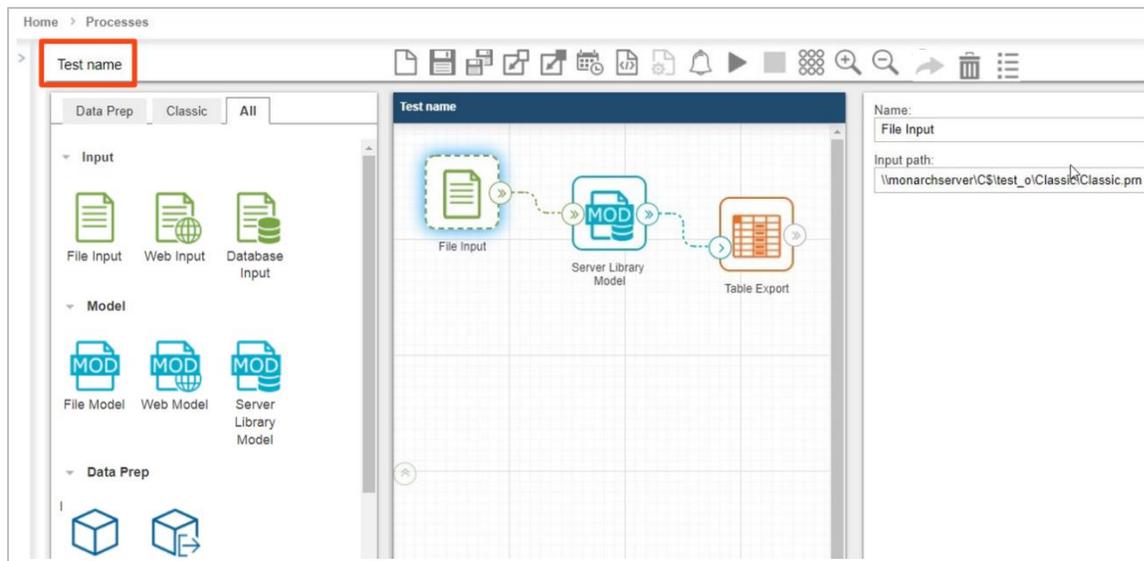
```

{
  "ProcessName": "Test name",
  "ProcessItems": [
    {
      "ItemType": "InputFile",
      "FileInput": {
        "FileLocation": "Input path",
        "Name": "File Input",
        "PdfPassword": "password"
      }
    }
  ]
}

```

To proceed with manifest monitoring

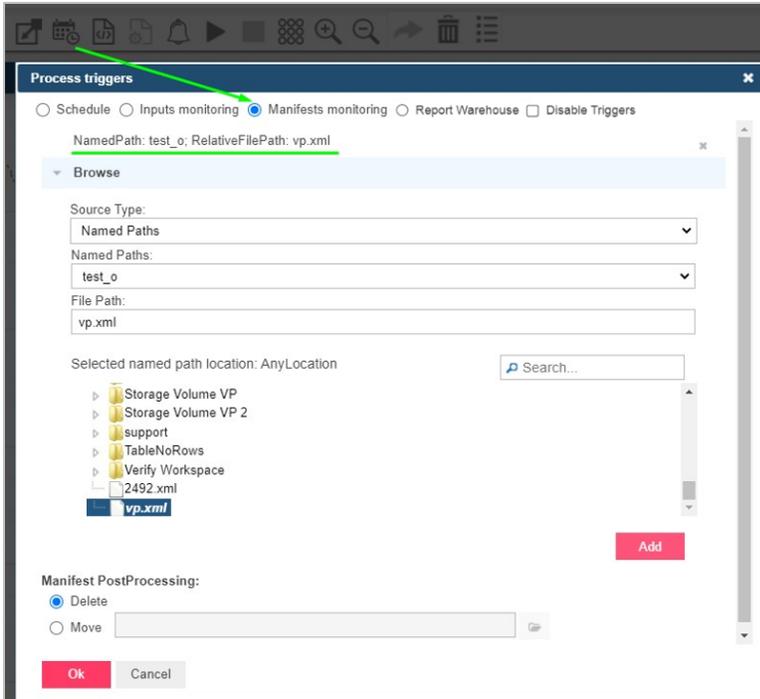
1. Create a new **Visual Process** (see [Processes](#)), and save it accordingly.



NOTE

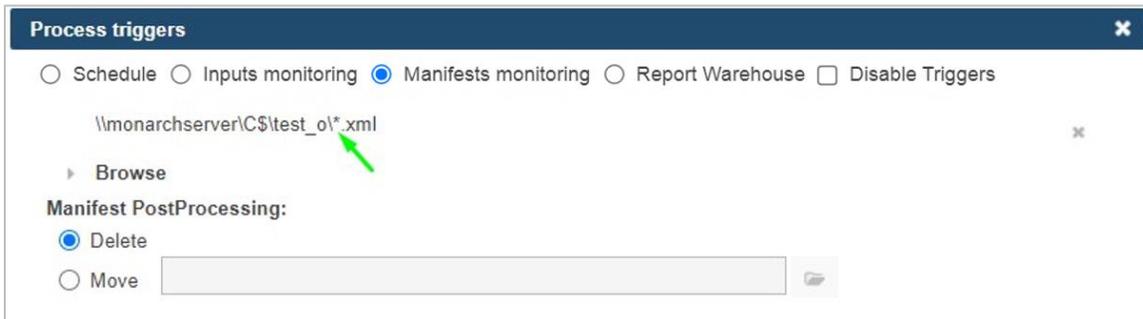
Different input files may be used in a process and in the XML file. For example, a Visual Process could have the input file “Classic.pn,” and the XML file could have the input file “Classic1.pn.”

2. Click the **Triggers** icon  to launch the **Process triggers** window.
3. Select **Manifests monitoring** (for details, see [Manifests Monitoring Trigger](#)).
 - a. Click **Browse**.
 - a. Select the **Source Type**, and add the manifest xml file you prepared (as described in the introduction of this section).
 - b. Click **Add**. The new manifest xml should appear on the list under **Manifests monitoring**.



NOTE You may use a wildcard (* or ?) to select any file (e.g., *.xml).

- c. Select one of two options in Manifest PostProcessing.
 - ◆ Delete (selected by default) for remove the XML file after processing
 - ◆ Move, set the folder path, for move the XML file after processing
- d. Click the **Ok** button to save the process monitoring.

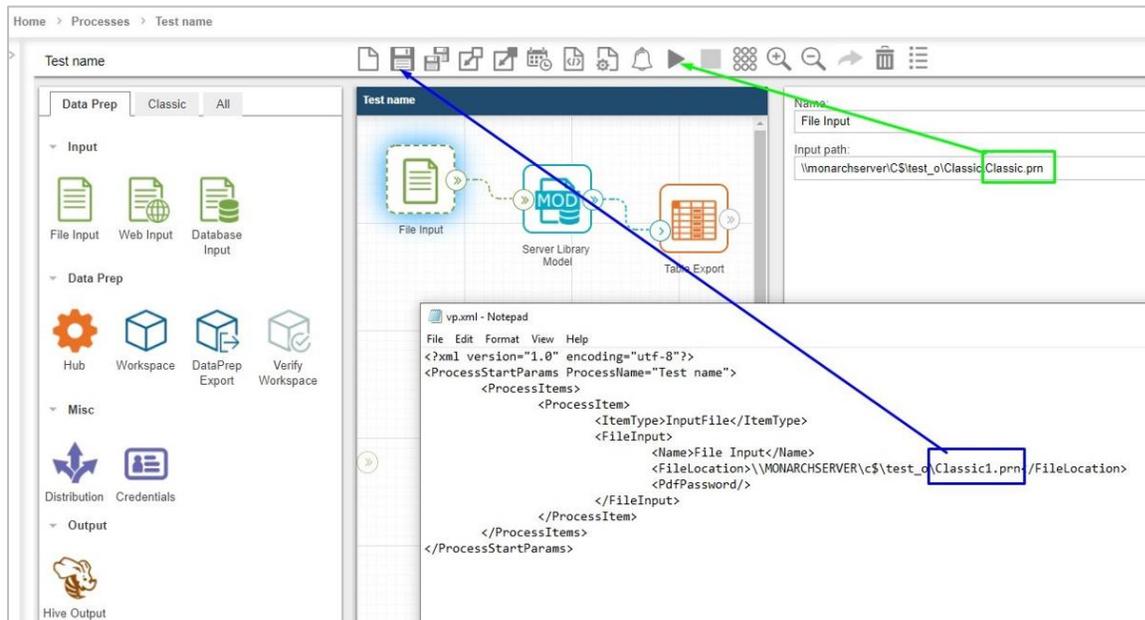


- 4. Save the process.

NOTE If you are using the input file specified in the xml file you created in the beginning

(*Classic1.prn* in the manifest xml example above), click the **Save** icon  only. When the xml file is found, the process will run automatically.

If you are using the general file input (Classic.prn), click the **Save and Run** icon  instead.



5. Close the process, and open **Logs**.

The log list should contain the process that will run.

The screenshot shows the 'Logs' page in the Monarch Server Automator. The left sidebar contains navigation options like 'My Home', 'Report Warehouse', 'Automator', 'Standard Processes', and 'Visual Processes'. The main area displays a table of logs with columns for LOG ID, PROCESS NAME, STARTED, COMPLETED, and DURATION. A log entry with LOG ID 2671 and PROCESS NAME 'Test name' is highlighted. Below the table, there are tabs for 'Text' and 'Visual', and a section for 'Process Events' and 'Process Items'.

LOG ID	PROCESS NAME	STARTED	COMPLETED	DURATION
2671	Test name	08/14/2020 4:37:16 PM	08/14/2020 4:37:21 PM	00:00:05
2668	Hub example for Lesson course 2020	08/11/2020 7:42:24 PM	08/11/2020 7:42:34 PM	00:00:10

The 'Process Events' section shows the following events:

- 8/14/2020, 7:37:16 PM Created.
- 8/14/2020, 7:37:16 PM Started by manifest \monarchserver\CS\test_o\vp.xml.
- 8/14/2020, 7:37:16 PM Will be processed on monarchserver.
- 8/14/2020, 7:37:21 PM The execution of the process is completed. Duration of 00:00:05.04.

The 'Process Items' section shows the following items:

- File Input** (2 Events, Duration 0.06(s))
 - 8/14/2020, 7:37:16 PM Started.
 - 8/14/2020, 7:37:16 PM Completed. Duration of 00:00:00.06
- Server Library Model** (2 Events, Duration 0.16(s))
- Table Export** (2 Events, Duration 0.45(s))
 - 8/14/2020, 7:37:16 PM Started.
 - 8/14/2020, 7:37:16 PM Completed. Duration of 00:00:00.45

The 'Table Export' item details are as follows:

Export Location	Table Name	Records	Description	Inputs
\monarchserver\CS\test_o\Exports\Classic_export_vp.csv	Classic1.prn	149	Export the table data, unfiltered, to a file.	\MONARCHSERVER\CS\test_o\Classic1.prn

6. Run the process again.

If you click on **Run now** or **Save and Run**, the general file input (Classic.prn) will be used.

NOTE The **manifest xml** file may only be run again in the following three cases:

- When changes are made to the xml file
- When a new manifest monitoring file is added to Triggers for a process
- When a new manifest monitoring file is added to the folder after using a wildcard (* or ?) in triggers

Under these cases, the manifest XML file will automatically run again.

XSD FOR STANDARD PROCESSES

Manifest XML for Standard Processes can be prepared using XSD schema, as shown below.

```
<?xml version="1.0" encoding="utf-8" ?>
<xs:schema attributeFormDefault="unqualified" elementFormDefault="qualified"
xmlns:xs="http://www.w3.org/2001/XMLSchema">
  <xs:element name="manifest">
    <xs:complexType>
```

```

<xs:sequence>
  <xs:element name="inputs" minOccurs="0">
    <xs:complexType>
      <xs:sequence>
        <xs:element name="item" minOccurs="0" maxOccurs="unbounded">
          <xs:complexType>
            <xs:sequence>
              <xs:element name="uri" type="xs:string" minOccurs="0" />
              <xs:element name="location" minOccurs="0">
                <xs:complexType>
                  <xs:simpleContent>
                    <xs:extension base="xs:string">
                      <xs:attribute name="temporary" type="xs:boolean" />
                    </xs:extension>
                  </xs:simpleContent>
                </xs:complexType>
              </xs:element>
              <xs:element name="db_import" minOccurs="0">
                <xs:complexType>
                  <xs:sequence>
                    <xs:element name="data_source" type="xs:string" minOccurs="0" />
                    <xs:element name="table_name" type="xs:string" minOccurs="0" />
                    <xs:element name="apply_filter" type="xs:string" minOccurs="0" />
                    <xs:element name="password" type="xs:string" minOccurs="0" />
                  </xs:sequence>
                  <xs:attribute name="excel_column_names_row" type="xs:boolean" />
                  <xs:attribute name="replace_nulls" type="xs:boolean" />
                </xs:complexType>
              </xs:element>
            </xs:sequence>
            <xs:attribute name="name" type="xs:string" use="optional" />
          </xs:complexType>
        </xs:element>
      </xs:sequence>
    </xs:complexType>
  </xs:element>
  <xs:element name="outputs" minOccurs="0">
    <xs:complexType>
      <xs:sequence>
        <xs:element name="item" minOccurs="0" maxOccurs="unbounded">
          <xs:complexType>
            <xs:sequence>
              <xs:element name="data_source" type="xs:string" minOccurs="0" />
              <xs:element name="table_name" type="xs:string" minOccurs="0" />
              <xs:element name="password" type="xs:string" minOccurs="0" />
            </xs:sequence>
            <xs:attribute name="name" type="xs:string" use="required" />
            <xs:attribute name="project" type="xs:string" use="required" />
          </xs:complexType>
        </xs:element>
      </xs:sequence>
    </xs:complexType>
  </xs:element>
  <xs:element name="projects" minOccurs="0">
    <xs:complexType>
      <xs:sequence>
        <xs:element name="add" minOccurs="0">
          <xs:complexType>
            <xs:sequence>
              <xs:element name="project" type="xs:string" minOccurs="0"
maxOccurs="unbounded"/>
            </xs:sequence>
          </xs:complexType>
        </xs:element>
        <xs:element name="remove" minOccurs="0">
          <xs:complexType>
            <xs:sequence>

```

```

        <xs:element name="project" type="xs:string" minOccurs="0"
maxOccurs="unbounded"/>
    </xs:sequence>
</xs:complexType>
</xs:element>
<xs:element name="order" minOccurs="0">
    <xs:complexType>
        <xs:sequence>
            <xs:element name="project" minOccurs="0" maxOccurs="unbounded">
                <xs:complexType>
                    <xs:simpleContent>
                        <xs:extension base="xs:string">
                            <xs:attribute name="order" type="xs:int" use="required" />
                        </xs:extension>
                    </xs:simpleContent>
                </xs:complexType>
            </xs:element>
        </xs:sequence>
    </xs:complexType>
</xs:element>
</xs:sequence>
</xs:complexType>
</xs:element>
<xs:element name="runtime_fields" minOccurs="0">
    <xs:complexType>
        <xs:sequence>
            <xs:element name="runtime_field" minOccurs="0" maxOccurs="unbounded">
                <xs:complexType>
                    <xs:simpleContent>
                        <xs:extension base="xs:string">
                            <xs:attribute name="name" type="xs:string" use="optional" />
                        </xs:extension>
                    </xs:simpleContent>
                </xs:complexType>
            </xs:element>
        </xs:sequence>
    </xs:complexType>
</xs:element>
<xs:attribute name="process" type="xs:string" use="optional" />
<xs:attribute name="sourceLocation" type="xs:string" use="optional" />
</xs:complexType>
</xs:element>
</xs:schema>

```

XSD FOR VISUAL PROCESSES

Manifest XML for Visual Processes can be prepared using XSD schema, as shown below.

```

<?xml version="1.0" encoding="utf-8" ?>
<xs:schema attributeFormDefault="unqualified" elementFormDefault="qualified"
xmlns:xs="http://www.w3.org/2001/XMLSchema">
    <xs:element name="ProcessStartParams">
        <xs:complexType>
            <xs:sequence>
                <xs:element name="ProcessItems" minOccurs="0" maxOccurs="1">
                    <xs:complexType>
                        <xs:sequence>
                            <xs:element name="ProcessItem" minOccurs="0" maxOccurs="unbounded">
                                <xs:complexType>
                                    <xs:sequence>
                                        <xs:element name="ItemType">

```

```

        <xs:simpleType>
          <xs:restriction base="xs:string">
            <xs:enumeration value="InputFile"></xs:enumeration>
            <xs:enumeration value="InputWeb"></xs:enumeration>
            <xs:enumeration value="InputDatabase"></xs:enumeration>
            <xs:enumeration value="InputContentSystem"></xs:enumeration>
            <xs:enumeration value="ModelFile"></xs:enumeration>
            <xs:enumeration value="ExportReport"></xs:enumeration>
            <xs:enumeration value="ExportTable"></xs:enumeration>
            <xs:enumeration value="ExportSummary"></xs:enumeration>
            <xs:enumeration value="DataPrepWorkspace"></xs:enumeration>
            <xs:enumeration value="DataPrepExport"></xs:enumeration>
            <xs:enumeration value="Credentials"></xs:enumeration>
            <xs:enumeration value="FileBurster"></xs:enumeration>
          </xs:restriction>
        </xs:simpleType>
      </xs:element>
      <xs:element name="FileInput" minOccurs="0" maxOccurs="1"
type="FileInput" />
      <xs:element name="WebInput" minOccurs="0" maxOccurs="1" type="WebInput"
/>
      <xs:element name="DatabaseInput" minOccurs="0" maxOccurs="1"
type="DatabaseInput" />
      <xs:element name="ContentSystemInput" minOccurs="0" maxOccurs="1"
type="ContentSystemInput" />
      <xs:element name="Model" minOccurs="0" maxOccurs="1" type="Model" />
      <xs:element name="DataPrepWorkspaceItem" minOccurs="0" maxOccurs="1"
type="DataPrepWorkspaceItem" />
      <xs:element name="DataPrepExportItem" minOccurs="0" maxOccurs="1"
type="DataPrepExportItem" />
      <xs:element name="ExportReport" minOccurs="0" maxOccurs="1"
type="ExportReport" />
      <xs:element name="ExportTable" minOccurs="0" maxOccurs="1"
type="ExportTable" />
      <xs:element name="ExportSummary" minOccurs="0" maxOccurs="1"
type="ExportSummary" />
      <xs:element name="Credentials" minOccurs="0" maxOccurs="1"
type="Credentials" />
      <xs:element name="FileBurster" minOccurs="0" maxOccurs="1"
type="FileBurster" />
    </xs:sequence>
  </xs:complexType>
</xs:element>
</xs:sequence>
</xs:complexType>
</xs:element>
<xs:element name="RuntimeFields" minOccurs="0">
  <xs:complexType>
    <xs:sequence>
      <xs:element name="RuntimeField" minOccurs="1" maxOccurs="unbounded">
        <xs:complexType>
          <xs:sequence>
            <xs:element name="Name" type="xs:string" />
            <xs:element name="Value" type="xs:string" />
          </xs:sequence>
        </xs:complexType>
      </xs:element>
    </xs:sequence>
  </xs:complexType>
</xs:element>
<xs:attribute name="ProcessName" type="xs:string" use="required" />
<xs:attribute name="SourceLocation" type="xs:string" use="optional" />
</xs:complexType>
</xs:element>

<xs:complexType name="ProcessItem">

```

```

    <xs:sequence>
      <xs:element name="Name" type="xs:string" />
    </xs:sequence>
  </xs:complexType>

  <xs:simpleType name="ExportTableExistence">
    <xs:restriction base="xs:string">
      <xs:enumeration value="Overwrite"></xs:enumeration>
      <xs:enumeration value="Append"></xs:enumeration>
      <xs:enumeration value="Update"></xs:enumeration>
      <xs:enumeration value="AppendAndUpdate"></xs:enumeration>
      <xs:enumeration value="Skip"></xs:enumeration>
    </xs:restriction>
  </xs:simpleType>

  <xs:simpleType name="ExportFileExistingOptions">
    <xs:restriction base="xs:string">
      <xs:enumeration value="Overwrite"></xs:enumeration>
      <xs:enumeration value="Append"></xs:enumeration>
      <xs:enumeration value="Skip"></xs:enumeration>
    </xs:restriction>
  </xs:simpleType>

  <xs:complexType name="Credential">
    <xs:sequence>
      <xs:element name="UserName" type="xs:string"></xs:element>
      <xs:element name="Password" type="xs:string"></xs:element>
      <xs:element name="Domain" type="xs:string"></xs:element>
      <xs:element name="UseCurrent" type="xs:boolean"></xs:element>
    </xs:sequence>
  </xs:complexType>

  <xs:complexType name="Credentials">
    <xs:complexContent>
      <xs:extension base="ProcessItem">
        <xs:sequence>
          <xs:element name="HttpCredential" type="Credential"></xs:element>
          <xs:element name="FtpCredential" type="Credential"></xs:element>
          <xs:element name="S3Credential" type="Credential"></xs:element>
        </xs:sequence>
      </xs:extension>
    </xs:complexContent>
  </xs:complexType>

  <xs:complexType name="FileInput">
    <xs:complexContent>
      <xs:extension base="ProcessItem">
        <xs:sequence>
          <xs:element name="FileLocation" type="xs:string" />
          <xs:element name="PdfPassword" type="xs:string" />
        </xs:sequence>
      </xs:extension>
    </xs:complexContent>
  </xs:complexType>

  <xs:complexType name="WebInput">
    <xs:complexContent>
      <xs:extension base="ProcessItem">
        <xs:sequence>
          <xs:element name="Location" type="xs:string" />
          <xs:element name="PdfPassword" type="xs:string" />
        </xs:sequence>
      </xs:extension>
    </xs:complexContent>
  </xs:complexType>

```

```

<xs:simpleType name="char">
  <xs:restriction base="xs:string">
    <xs:length fixed="true" value="1"/>
  </xs:restriction>
</xs:simpleType>
<xs:complexType name="DatabaseInput">
  <xs:complexContent>
    <xs:extension base="ProcessItem">
      <xs:sequence>
        <xs:element name="DataSource" type="xs:string" />
        <xs:element name="Password" type="xs:string" />
        <xs:element name="TableName" type="xs:string" />
        <xs:element name="Filter" type="xs:string" />
        <xs:element name="ReplaceNulls" type="xs:boolean" nillable="true" />
        <xs:element name="FirstRowContainsNames" type="xs:boolean" nillable="true" />
        <xs:element name="TextParameters">
          <xs:complexType>
            <xs:sequence>
              <xs:element name="LinesToIgnoreAtStart" type="xs:int"
nillable="true"/></xs:element>
              <xs:element name="AllowEmbeddedLineBreaks" type="xs:boolean"
nillable="true"/></xs:element>
              <xs:element name="Delimiter" type="char" />
              <xs:element name="TextQualifier" type="char" />
              <xs:element name="TextEncoding" nillable="true">
                <xs:simpleType>
                  <xs:restriction base="xs:string">
                    <xs:enumeration value="Ansi"/></xs:enumeration>
                    <xs:enumeration value="Ascii"/></xs:enumeration>
                    <xs:enumeration value="Utf8"/></xs:enumeration>
                    <xs:enumeration value="Utf16Le"/></xs:enumeration>
                    <xs:enumeration value="Utf16Be"/></xs:enumeration>
                    <xs:enumeration value="Default"/></xs:enumeration>
                  </xs:restriction>
                </xs:simpleType>
              </xs:element>
            </xs:sequence>
          </xs:complexType>
        </xs:element>
      </xs:sequence>
    </xs:extension>
  </xs:complexContent>
</xs:complexType>

<xs:complexType name="ContentSystemInput">
  <xs:complexContent>
    <xs:extension base="ProcessItem">
      <xs:sequence>
        <xs:element name="Since" type="xs:string" nillable="true" />
        <xs:element name="Till" type="xs:string" nillable="true" />
        <xs:element name="DocType" >
          <xs:complexType>
            <xs:sequence>
              <xs:element name="Id" type="xs:int"/></xs:element>
            </xs:sequence>
          </xs:complexType>
        </xs:element>
        <xs:element name="SelectCriteria">
          <xs:simpleType>
            <xs:restriction base="xs:string">
              <xs:enumeration value="ByDate"/>
              <xs:enumeration value="Newest"/>
            </xs:restriction>
          </xs:simpleType>
        </xs:element>
        <xs:element name="DateType">
          <xs:simpleType>

```

```

        <xs:restriction base="xs:string">
            <xs:enumeration value="FilingDate"/>
            <xs:enumeration value="DocumentDate"/>
        </xs:restriction>
    </xs:simpleType>
</xs:element>
</xs:sequence>
</xs:extension>
</xs:complexContent>
</xs:complexType>

<xs:complexType name="Model">
    <xs:complexContent>
        <xs:extension base="ProcessItem">
            <xs:sequence>
                <xs:element name="Location" type="xs:string" />
                <xs:element name="ExternalLookupOptions" minOccurs="0" maxOccurs="1">
                    <xs:complexType>
                        <xs:sequence>
                            <xs:element name="ExternalLookupOption" maxOccurs="unbounded"
minOccurs="0">
                                <xs:complexType>
                                    <xs:sequence>
                                        <xs:element name="Name" type="xs:string" />
                                        <xs:element name="Password" type="xs:string" />
                                    </xs:sequence>
                                </xs:complexType>
                            </xs:element>
                        </xs:sequence>
                    </xs:complexType>
                </xs:element>
            </xs:sequence>
        </xs:extension>
    </xs:complexContent>
</xs:complexType>

<xs:complexType name="DataPrepWorkspaceItem">
    <xs:complexContent>
        <xs:extension base="ProcessItem">
            <xs:sequence>
                <xs:element name="WorkspacePath" type="xs:string"></xs:element>
                <xs:element name="LoadPlans" minOccurs="0" maxOccurs="1">
                    <xs:complexType>
                        <xs:sequence>
                            <xs:element name="LoadPlan" minOccurs="0" maxOccurs="unbounded">
                                <xs:complexType>
                                    <xs:sequence>
                                        <xs:element name="TableName" type="xs:string" />
                                        <xs:element name="LoadPlanName" type="xs:string" />
                                    </xs:sequence>
                                </xs:complexType>
                            </xs:element>
                        </xs:sequence>
                    </xs:complexType>
                </xs:element>
            </xs:sequence>
        </xs:extension>
    </xs:complexContent>
</xs:complexType>

<xs:complexType name="DpFileExportOptionsBase">
    <xs:sequence>

```

```

        <xs:element name="Destination" type="xs:string"></xs:element>
        <xs:element name="OverwriteFile" type="xs:boolean" nillable="true"></xs:element>
        <xs:element name="WhenFileExist" type="ExportFileExistingOptions"
nillable="true"></xs:element>
    </xs:sequence>
</xs:complexType>
<xs:complexType name="DpDbExportOptionsBase">
    <xs:complexContent>
        <xs:extension base="DpFileExportOptionsBase">
            <xs:sequence>
                <xs:element name="TableName" type="xs:string"></xs:element>
                <xs:element name="WhenTableExist" type="ExportTableExistence"
nillable="true"></xs:element>
            </xs:sequence>
        </xs:extension>
    </xs:complexContent>
</xs:complexType>
<xs:complexType name="DataPrepExportItem">
    <xs:complexContent>
        <xs:extension base="ProcessItem">
            <xs:sequence>
                <xs:element name="ExportPlanName" type="xs:string"></xs:element>
                <xs:element name="AllLoadPlans" type="xs:boolean" nillable="true"></xs:element>
                <xs:element name="ExportPlanNames" minOccurs="0" maxOccurs="unbounded"
type="xs:string"></xs:element>
                <xs:element name="ExportType">
                    <xs:simpleType>
                        <xs:restriction base="xs:string">
                            <xs:enumeration value="Delimited"></xs:enumeration>
                            <xs:enumeration value="MicrosoftExcel"></xs:enumeration>
                            <xs:enumeration value="MicrosoftAccess"></xs:enumeration>
                            <xs:enumeration value="DatawatchDesigner"></xs:enumeration>
                            <xs:enumeration value="Qlik"></xs:enumeration>
                            <xs:enumeration value="Tableau"></xs:enumeration>
                            <xs:enumeration value="OleDb"></xs:enumeration>
                            <xs:enumeration value="Angoss"></xs:enumeration>
                            <xs:enumeration value="Cognos"></xs:enumeration>
                            <xs:enumeration value="PowerBI"></xs:enumeration>
                            <xs:enumeration value="Fixed"></xs:enumeration>
                            <xs:enumeration value="Sas Transport Format"></xs:enumeration>
                            <xs:enumeration value="Swarm"></xs:enumeration>
                        </xs:restriction>
                    </xs:simpleType>
                </xs:element>
                <xs:element name="DelimitedExportOptions">
                    <xs:complexType >
                        <xs:complexContent>
                            <xs:extension base="DpFileExportOptionsBase">
                                <xs:sequence>
                                    <xs:element name="IncludeHeader" type="xs:boolean"
nillable="true"></xs:element>
                                    <xs:element name="Delimiter" type="xs:string"></xs:element>
                                    <xs:element name="Qualifier" type="xs:string"></xs:element>
                                </xs:sequence>
                            </xs:extension>
                        </xs:complexContent>
                    </xs:complexType>
                </xs:element>
                <xs:element name="QlikExportOptions">
                    <xs:complexType >
                        <xs:complexContent>
                            <xs:extension base="DpFileExportOptionsBase">
                                </xs:extension>
                            </xs:complexContent>
                        </xs:complexType>
                    </xs:element>
                <xs:element name="AngossExportOptions">

```

```

<xs:complexType >
  <xs:complexContent>
    <xs:extension base="DpFileExportOptionsBase">
    </xs:extension>
  </xs:complexContent>
</xs:complexType>
</xs:element>
<xs:element name="SasTransportExportOptions">
  <xs:complexType >
    <xs:complexContent>
      <xs:extension base="DpDbExportOptionsBase">
      </xs:extension>
    </xs:complexContent>
  </xs:complexType>
</xs:element>
<xs:element name="TableauExportOptions">
  <xs:complexType>
    <xs:complexContent>
      <xs:extension base="DpFileExportOptionsBase">
      </xs:extension>
    </xs:complexContent>
  </xs:complexType>
</xs:element>
<xs:element name="ExcelExportOptions">
  <xs:complexType>
    <xs:complexContent>
      <xs:extension base="DpDbExportOptionsBase">
      </xs:extension>
    </xs:complexContent>
  </xs:complexType>
</xs:element>
<xs:element name="DesignerExportOptions">
  <xs:complexType>
    <xs:complexContent>
      <xs:extension base="DpDbExportOptionsBase">
      </xs:extension>
    </xs:complexContent>
  </xs:complexType>
</xs:element>
<xs:element name="AccessExportOptions">
  <xs:complexType>
    <xs:complexContent>
      <xs:extension base="DpDbExportOptionsBase">
      </xs:extension>
    </xs:complexContent>
  </xs:complexType>
</xs:element>
<xs:element name="PowerBIExportOptions">
  <xs:complexType>
    <xs:complexContent>
      <xs:extension base="DpDbExportOptionsBase">
        <xs:sequence>
          <xs:element name="AccessToken" type="xs:string"></xs:element>
          <xs:element name="RefreshToken" type="xs:string"></xs:element>
        </xs:sequence>
      </xs:extension>
    </xs:complexContent>
  </xs:complexType>
</xs:element>
<xs:element name="CognosExportOptions">
  <xs:complexType>
    <xs:complexContent>
      <xs:extension base="DpDbExportOptionsBase">
        <xs:sequence>
          <xs:element name="ServerUrl" type="xs:string"></xs:element>
          <xs:element name="Namespace" type="xs:string"></xs:element>
          <xs:element name="UserName" type="xs:string"></xs:element>
        </xs:sequence>
      </xs:extension>
    </xs:complexContent>
  </xs:complexType>
</xs:element>

```

```

        <xs:element name="Password" type="xs:string"></xs:element>
        <xs:element name="IsAnonymous" type="xs:boolean"></xs:element>
    </xs:sequence>
</xs:extension>
</xs:complexContent>
</xs:complexType>
</xs:element>
<xs:element name="OleDbExportOptions">
    <xs:complexType>
        <xs:sequence>
            <xs:element name="IncludePassword" type="xs:boolean"
nillable="true"></xs:element>
            <xs:element name="Password" type="xs:string"></xs:element>
            <xs:element name="TableName" type="xs:string"></xs:element>
            <xs:element name="WhenTableExist" type="ExportTableExistence"
nillable="true"></xs:element>
            <xs:element name="Destination" type="xs:string"></xs:element>
        </xs:sequence>
    </xs:complexType>
</xs:element>
<xs:element name="FixedExportOptions">
    <xs:complexType>
        <xs:sequence>
            <xs:element name="Destination" type="xs:string"></xs:element>
            <xs:element name="WhenFileExist" type="ExportFileExistingOptions"
nillable="true"></xs:element>
        </xs:sequence>
    </xs:complexType>
</xs:element>
<xs:element name="SwarmExportOptions">
    <xs:complexType>
        <xs:sequence>
            <xs:element name="TableName" type="xs:string"></xs:element>
            <xs:element name="DestinationFolderId" type="xs:long"
nillable="true"></xs:element>
            <xs:element name="UserName" type="xs:string"></xs:element>
            <xs:element name="Password" type="xs:string"></xs:element>
            <xs:element name="UseDefaultCredentials" type="xs:boolean"
nillable="true"></xs:element>
        </xs:sequence>
    </xs:complexType>
</xs:element>
</xs:sequence>
</xs:extension>
</xs:complexContent>
</xs:complexType>

<xs:complexType name="ExportReport">
    <xs:complexContent>
        <xs:extension base="ProcessItem">
            <xs:sequence>
                <xs:element name="Destination" type="xs:string"></xs:element>
                <xs:element name="ExportFileExistingOptions" type="ExportFileExistingOptions"
nillable="true"></xs:element>
            </xs:sequence>
        </xs:extension>
    </xs:complexContent>
</xs:complexType>

<xs:simpleType name="AutoNamingMethod">
    <xs:restriction base="xs:string">
        <xs:enumeration value="Unspecified"></xs:enumeration>
        <xs:enumeration value="Summary"></xs:enumeration>
        <xs:enumeration value="Measure"></xs:enumeration>
        <xs:enumeration value="Split"></xs:enumeration>
    </xs:restriction>
</xs:simpleType>

```

```

    <xs:enumeration value="Filter"></xs:enumeration>
  </xs:restriction>
</xs:simpleType>

<xs:complexType name="ExportTable">
  <xs:complexContent>
    <xs:extension base="ProcessItem">
      <xs:sequence>
        <xs:element name="OutputType">
          <xs:simpleType>
            <xs:restriction base="xs:string">
              <xs:enumeration value="FileSystem"></xs:enumeration>
              <xs:enumeration value="Database"></xs:enumeration>
            </xs:restriction>
          </xs:simpleType>
        </xs:element>
        <xs:element name="Destination" type="xs:string"></xs:element>
        <xs:element name="FileName" type="xs:string"></xs:element>
        <xs:element name="ExportFileType" type="xs:string"></xs:element>
        <xs:element name="WhenFileExist" type="ExportFileExistingOptions"
nillable="true"></xs:element>
        <xs:element name="ConnectionString" type="xs:string"></xs:element>
        <xs:element name="ConnectionStringPassword" type="xs:string"></xs:element>
        <xs:element name="TableName" type="xs:string"></xs:element>
        <xs:element name="WhenTableExist" type="ExportTableExistence"
nillable="true"></xs:element>
        <xs:element name="ByFilesNaming" type="AutoNamingMethod"
nillable="true"></xs:element>
        <xs:element name="ByTablesNaming" type="AutoNamingMethod"
nillable="true"></xs:element>
        <xs:element name="CurrentSort" maxOccurs="1">
          <xs:complexType>
            <xs:sequence>
              <xs:element name="Name" type="xs:string"></xs:element>
              <xs:element name="NoSort" type="xs:boolean"></xs:element>
            </xs:sequence>
          </xs:complexType>
        </xs:element>
        <xs:element name="CurrentFilter" maxOccurs="1">
          <xs:complexType>
            <xs:sequence>
              <xs:element name="Name" type="xs:string"></xs:element>
              <xs:element name="NoFilter" type="xs:boolean"></xs:element>
              <xs:element name="AllFilters" type="xs:boolean"></xs:element>
            </xs:sequence>
          </xs:complexType>
        </xs:element>
      </xs:sequence>
    </xs:extension>
  </xs:complexContent>
</xs:complexType>

<xs:complexType name="ExportSummary">
  <xs:complexContent>
    <xs:extension base="ExportTable">
      <xs:sequence>
        <xs:element name="CurrentSummary" maxOccurs="1">
          <xs:complexType>
            <xs:sequence>
              <xs:element name="Name" type="xs:string"></xs:element>
              <xs:element name="AllSummaries" type="xs:boolean"></xs:element>
            </xs:sequence>
          </xs:complexType>
        </xs:element>
      </xs:sequence>
    </xs:extension>
  </xs:complexContent>
</xs:complexType>

```

```

</xs:complexType>

<xs:simpleType name="BurstType">
  <xs:restriction base="xs:string">
    <xs:enumeration value="Defined"></xs:enumeration>
    <xs:enumeration value="Pages"></xs:enumeration>
  </xs:restriction>
</xs:simpleType>
<xs:simpleType name="CanNotBurstFile">
  <xs:restriction base="xs:string">
    <xs:enumeration value="Skip"></xs:enumeration>
    <xs:enumeration value="Stop"></xs:enumeration>
  </xs:restriction>
</xs:simpleType>

<xs:complexType name="DefinitionBurstOptions">
  <xs:sequence>
    <xs:element name="IndexTableStartMarker" type="xs:string"></xs:element>
    <xs:element name="LineValidator" type="xs:string"></xs:element>
    <xs:element name="TableEndMarker" type="xs:string"></xs:element>
    <xs:element name="ContentStartMarker" type="xs:string"></xs:element>

    <xs:element name="IndexTableStartMarkerOffset" type="xs:int"
nillable="true"></xs:element>
    <xs:element name="LineValidatorOffset" type="xs:int" nillable="true"></xs:element>
    <xs:element name="LineFieldOffset" type="xs:int" nillable="true"></xs:element>
    <xs:element name="LineFieldLength" type="xs:int" nillable="true"></xs:element>
    <xs:element name="IdFieldOffset" type="xs:int" nillable="true"></xs:element>
    <xs:element name="IdFieldLength" type="xs:int" nillable="true"></xs:element>
    <xs:element name="TableEndMarkerOffset" type="xs:int" nillable="true"></xs:element>
    <xs:element name="ContentStartMarkerOffset" type="xs:int"
nillable="true"></xs:element>
  </xs:sequence>
</xs:complexType>
<xs:complexType name="PageBurstOptions">
  <xs:sequence>
    <xs:element name="ModelPath" type="xs:string"></xs:element>
    <xs:element name="PagesPerOutputFile" type="xs:int" nillable="true"></xs:element>
    <xs:element name="ProcessSuccessiveFilesAsOne" type="xs:boolean"
nillable="true"></xs:element>
  </xs:sequence>
</xs:complexType>

<xs:complexType name="FileBurster">
  <xs:complexContent>
    <xs:extension base="ProcessItem">
      <xs:sequence>
        <xs:element name="DestinationPath" type="xs:string"></xs:element>
        <xs:element name="OutputFileNameMask" type="xs:string"></xs:element>
        <xs:element name="UseHeaderOutputMask" type="xs:boolean"
nillable="true"></xs:element>
        <xs:element name="HeaderOutputMask" type="xs:string"></xs:element>
        <xs:element name="ErrorDestination" type="xs:string"></xs:element>

        <xs:element name="UseRecognizerLineKeyMask" type="xs:boolean"
nillable="true"></xs:element>
        <xs:element name="RecognizerLineKeyMask" type="xs:string"></xs:element>
        <xs:element name="RecognizerLineKeyMaskOffset" type="xs:int"
nillable="true"></xs:element>

        <xs:element name="IsTrimLeadingCharacters" type="xs:boolean"
nillable="true"></xs:element>
        <xs:element name="TrimLeadingCharacters" type="xs:int"
nillable="true"></xs:element>

        <xs:element name="BurstType" type="BurstType" nillable="true"></xs:element>

```

```
        <xs:element name="DefinitionBurstOptions"
type="DefinitionBurstOptions"></xs:element>
        <xs:element name="PageBurstOptions" type="PageBurstOptions"></xs:element>
    </xs:sequence>
</xs:extension>
</xs:complexContent>
</xs:complexType>

</xs:schema>
```

RMS

The topics in this section provide information applicable to the Report Mining Server application and deal with:

- ❑ [Content Recognition Rules](#)
- ❑ [Creating or Editing Content Recognition Rules](#)

CONTENT RECOGNITION RULES

Content recognition rules allow to automatically load models based on the content of the reports uploaded by the user. This relieves the user of some routine operations.

When a report is uploaded, it is searched for key words, and, in case of a match, a corresponding model is loaded.

Creating or Editing Content Recognition Rules

The **Content Recognition Rule** dialog provides a convenient and easy way of creating a new or editing an existing **Content Recognition Rule**.

All the Content Recognition Rules are listed on the [Content Recognition Rules](#) page. To display it, click **RMS** on the Navigation Tree, then click **Content Recognition Rules**.

To add a new content recognition rule, click **Add New Content Recognition Rule** icon  above or below the list of content recognition rules.

To delete a content recognition rule, select it, and click the **Delete** icon . You can also click the  icon next to a rule to delete it.

To edit a content recognition rule, select it, and click the Edit icon .

Content Recognition Rule

Name:

Max Pages To Scan:

Criteria:

Use wildcard Case sensitive

Models:

[Show advanced options](#)

The following fields should be filled to create a **Content Recognition Rule**:

- ❑ **Name:** The name of your rule.
- ❑ **Description:** The description of your rule, this field is hidden, to show it click Show advanced options. This field is not mandatory.

- ❑ **Max Pages to Scan:** The maximum number of page that will be searched for a pattern specified in the Criteria field.
- ❑ **Criteria:** Enter the word or phrase that you want the program to recognize. You can use wildcard characters * (to substitute for any zero or more characters) and ? (to substitute for a single character), in case you do, activate the Use wildcard check box under the Criteria field. To make the search case sensitive, activate the Case sensitive check box.
- ❑ **Models:** Lists the models that will be loaded in case of Criteria match. Clicking Add Model opens a model selection dialog.

Clicking **Save** closes the dialog and creates the rule. Clicking **Clear** clears the fields. **Cancel** closes the dialog without saving the changes.

PORTLETS

A portlet contains one or more visual elements used for presentation of data. The Portlets module allows the integration of portlets — Panopticon Workbooks — with Monarch Server. Users of MSCClient and RMSClient can view Panopticon Workbooks in the application according to their access rights and privileges.

Portlets are created and managed in MSAdmin. For more information, see [Portlets](#).

For convenience, portlets are organized into [portlet groups](#).

PORTLETS

You can [add](#), [edit](#), and [delete](#) portlets. To make portlets available to users, you assign the appropriate portlet access rights to users, user groups, or roles.

All available portlets are listed on the Portlets page. To display it, on the Navigation Tree, click **Portlets**, and then select **Portlets**.

Adding a Portlet

To add a portlet

1. On the Navigation Tree, click **Portlets**, and then select **Portlets**.
2. Click  on the Portlets page. The **Portlet** page appears.
3. Enter a unique name of the new portlet in the **Name** field.
4. Enter any text defining the portlet in the **Description** field. This step is optional.
5. Enter the full URL to the portlet item on the Panopticon Workbooks in the **Url** field.

NOTE This URL is made by combining the base URL of the portlet group to which the portlet is added and the URL of the portlet itself (i.e. the name of the workbook). Therefore, you can also specify only a unique name that identifies the workbook.

6. To save the new portlet and close the page, click **Save**.
7. The system saves the new portlet to the database and displays the updated **Portlets** page.
8. To save the new portlet and open it immediately in the editing mode for assigning portlet access rights, click the **Save/Edit** button. For more information on how to assign portlet access rights, see Steps 4-8 in "[Editing a Portlet](#)".
9. To clear all the fields, click **Clear**.
10. To close the **Portlet** page without saving, click **Cancel**. Keep in mind that no changes will be saved

NOTE After you have added the portlet, you have to assign access rights for it (see [Editing a Portlet](#), steps 4-8), as well as [assign it to a portlet group](#).

Editing a Portlet

To edit a portlet

1. On the Navigation Tree, click **Portlets**, and select **Portlets**.
2. Click the name of the portlet you want to edit or click anywhere on the row, and then click .
3. Make the necessary changes on the **Portlet** tab if required. For details about how to change portlet properties, refer to [Adding a Portlet](#).
4. Click the **Rights** tab to assign portlet access rights to users, user groups, or roles.

NOTE When **Global** check box is selected the portlet is available to all users. The access rights to specific portlet can also be granted on the corresponding **User**, **User Group**, or **Role** page. For more information, see [Assigning Access Rights](#).

5. Click **User Groups** to expand a hidden section. Select user groups that you want to view the portlet.
 - a. To add a user group drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a user group drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

6. Click **Roles** to expand a hidden section. Select roles that you want to view the portlet.
 - a. To add a role drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a role drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

7. In the **Users** section, select users that you want to view the dashboard.
 - a. To add a user drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a user drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

8. To save the changes, click **Save**.

NOTE For the portlet to be visible in MSCClient and RMSClient to a user with the portlet access rights, the user must also have the Portlets [privilege](#) enabled at the user, user group, or role level.

Deleting a Portlet

To delete a portlet

1. On the Navigation Tree, click **Portlets**, and then select **Portlets**.
2. Click **✕** at the end of the row with the portlet you want to delete. A dialog box appears.
3. Click **Delete** to delete the portlet from the list.

PORTLET GROUPS

Users of MSCClient and RMSClient see portlets organized into logical groupings — portlet groups — and arranged according to a particular layout.

Portlet groups are managed in MSAdmin. All available portlet groups are listed on the Portlet Groups page. To display it, on the Navigation Tree, click **Portlets**, and then select **Portlet Groups**.

Use this page to [add](#) portlet groups, assign portlets to them, and set their layouts. You can also [edit](#) and [delete](#) existing portlet groups.

Adding a Portlet Group

To add a portlet group

1. On the Navigation Tree, click **Portlets**, and then select **Portlet Groups**.
2. Click **+** on the **Portlet Groups** page. The **Portlet Group** page appears.
3. **Name:** Enter a unique name of the new portlet.
4. **Description:** Optionally, enter any text defining the portlet.
5. **Base Url:** Provide the base URL of the portlets belonging to the current group.

NOTE If you have already provided the full URLs for the portlets of the group, you can leave the **Base Url** field empty.

6. **Layout size:** Select a layout in which the portlets will be displayed for the user in the Server client application. Possible layouts range from 1x1 (1 item arranged horizontally x 1 item vertically) to 4x4 (4 items arranged horizontally x 4 items vertically).

7. **Available Portlets:** Contains the list of portlets from which you select those to be assigned to the current portlet group. To assign a portlet to the group, drag it from the left pane to the right pane or click the plus sign next to the portlet name.
8. **Assigned Portlets:** Contains the portlets that you assign to the current group. To remove a portlet from the group, drag it from the right pane to the left pane or click the minus sign next to the portlet name.

NOTE The maximum number of portlets that can be assigned to a group is 16, which is limited by the maximum layout size 4x4.

9. **Add All:** Click to assign all available portlets to the current group.
10. **Remove All:** Click to remove all assigned portlets from the current group.
11. To save the new portlet group, click the **Save** button.
The system saves the new portlet group to the database and displays the updated **Portlet Groups** page.
12. To reset the values, click the **Clear** button.
13. To close the **Portlet Groups** page without saving, click **Cancel**. Keep in mind that no changes will be saved.

Editing a Portlet Group

To edit a portlet group

1. On the Navigation Tree, click **Portlets**, and select **Portlet Groups**.
2. Click the name of the portlet group you want to edit or clicking anywhere on its row, and then click .
3. Make the necessary changes. For more information on how to change portlet group properties, see [Adding a Portlet Group](#).
4. Click **Save**.
The system saves the modified portlet group to the database and displays the updated **Portlet Groups** page.

Deleting a Portlet Group

To delete a portlet group

1. On the Navigation Tree, click **Portlets**, and then select **Portlet Groups**.
2. Click  at the end of the row with the portlet group you want to delete. A dialog box appears.
3. Click **Delete** to delete the portlet group from the list.

SERVER LIBRARY

The topics in this section provide information about:

- [Folders](#)
- [Project Files](#)
- [Workspaces](#)
- [Change Lists](#)
- [Models](#)
- [Templates](#)
- [Bulk Edits](#)

As a Monarch Server Report Mining Edition administrator you can create server library folders that include particular models and templates.

Users that indicate particular server library folder name when they connect to RMS client will be able to use the models and templates assigned to it and its subfolders.

FOLDERS

You can create server library folders that include particular project, models and templates.

Users that indicate particular server library folder name when they connect to RMS client will be able to use the models and templates assigned to it and its subfolders.

You can use this page to do the following:

- [Add a folder](#)
- [Edit a folder](#)
- [Delete a folder](#)

Adding a Folder

To add a folder

1. On the Navigation Tree, click **Server Library** and select **Folders**.
2. Click  on the **Folders** page. The **Folders** page appears.
3. Enter the name of a folder in the **Name** field.
4. Enter the description of a folder in the **Description** field. This step is optional.
5. In the **Available Models** section select models that you want the user to have access to.
 - a. To add a model drag it from the left pane to the right pane or click  icon next to its name.
 - b. To remove a model drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

6. In the **Available Projects** section select projects that you want the user to have access to.
 - a. To add a project drag it from the left pane to the right pane or click  icon next to its name.
 - b. To remove a project drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

7. In the **Available Templates** section select templates that you want the user to have access to.
 - a. To add a template drag it from the left pane to the right pane or click  icon next to its name.
 - b. To remove a template drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

8. In the **Available Workspaces** section select workspaces that you want the user to have access to.
 - a. To add a workspace drag it from the left pane to the right pane or click  icon next to its name.
 - b. To remove a workspace drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

9. In the **Available Change Lists** section select change lists that you want the user to have access to.
 - a. To add a change list drag it from the left pane to the right pane or click  icon next to its name.
 - b. To remove a change list drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

10. Click **Save** to save all settings done.

You can build server library folders hierarchy. Inside a folder of the root level, there might be several levels of enclosed or nested folders.

To add a nested folder

1. Open the required folder of the root level on the on the **Folders** page.
2. Enter the name of a folder in the **Name** field.

3. Define the level of the nested folder in the **Hierarchy level** tree.
4. Enter the description of a folder in the **Description** field. This step is optional.
5. Perform actions for available projects, models, templates, specified in the previous procedure (Adding a folder).
6. Click **Save** to save all settings of the nested folder.

When server library folders hierarchy created, on the dialogs for selecting projects, models, or templates, all these models will be displayed as a tree.

Editing a Folder

To edit a folder

1. On the Navigation Tree, click **Server Library** and select **Folders**.
2. Click the name of the folder you want to edit or click anywhere on its row, and then click .
3. Make the necessary changes. For details about how to change group parameters refer to [Adding a Folder](#).
4. To the changes, click the **Save** button.
The system saves the modified model to the database and displays the updated **Folders** page, if the operation is completed successfully.

To assign folder to user groups, roles, or users

1. On the Navigation Tree, click **Server Library** and select **Folders**.
2. Select the folder you want to edit on the **Folders** page. Click the name of the folder you want to edit or click anywhere on the row, and then click .
3. Click the **Rights** tab. For more information, see [Assigning Folder Access Rights](#).

Assigning Folder Access Rights

The creator of a folder automatically receives the rights to view and edit it.

You can assign folder access rights to other users. Authorized users can view and edit folders assigned to them in MSAdmin.

To assign folder access rights

1. On the Navigation Tree, click **Server Library** and select **Folders**.
2. Click the name of the folder you want to assign access rights for.
3. On the displayed page, select the **Rights** tab.
4. Select **Global** check box for the folder to be available to all users and click the **Save** button.
Otherwise, proceed to the next steps.
5. Click **User Groups** to expand a hidden section. Select the user groups that you want to have access to the folder.
 - a. To add a user group, drag it from the left pane to the right pane or click  icon next to its name.
 - b. To remove a user group, drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

6. Click **Roles** to expand a hidden section. Select the roles that you want to have access to the folder.
 - a. To add a role, drag it from the left pane to the right pane or click  icon next to its name.
 - b. To remove a role, drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

7. Click **Users** to expand a hidden section. Select the users that you want to have access to the folder.
 - a. To add a user, drag it from the left pane to the right pane or click  icon next to its name.
 - b. To remove a user, drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

8. To apply the changes, click the **Save** button.

NOTE If you don't specify any access rights for a folder, then it will be available for all users.

Deleting a Folder

To delete a folder from the list

1. On the Navigation Tree, click **Server Library** and select **Folders**.
2. Click  at the end of the row with the folder you want to delete. A dialog box appears.
3. Click **Delete** to delete the folder from the list.

PROJECTS

Project files are special files created with Project Editor (or with Monarch) that have a .xprj or .dprj extension and which contain all of the information necessary to execute one or more Automator operations. Projects utilize text or database input, extract data from the input via Monarch model files, and export the extracted data to various file types (for e, XLS, PDF, MDB, TXT, HTML, XML, etc.).

Projects are a key component of processes as when you [create a process in Automator](#), one of the key steps you perform is to assign a project (or projects) to it.

With Automator you can use an existing project file created with Monarch or you can [create a new project file](#) via the Project Editor.

All project files used in the system are displayed on the **Projects** page. To view this page, on the Navigation Tree, click **Server Library** and select **Projects**.

You can use the **Projects** page to do the following:

- ❑ [Add an existing project file created with Monarch](#)
- ❑ [Edit a project file \(name, revision, access rights\)](#)
- ❑ [Open a project file in the Project Editor](#)
- ❑ [Delete a project file from the system](#)
- ❑ [Change a project file state](#)

Adding a Project File

To add an existing project file created with Monarch

1. On the Navigation Tree, click **Server Library**, and select **Projects**.
2. On the displayed page, click the **Add** icon (). The **Project** page appears.
3. On the **Project** page, fill out the following fields:
 - **Name:** Enter the name of a project file.
 - **Description:** An optional description of a project file.
 - **Project:** Displays the path to a project file, after you click the Browse button and select a model file.
 - **Comment:** Enter any comments if necessary.
 - **Assign to Folders:** Adds the project file to a new or existing folder. To add the project file to a new folder, click **Create New Folder**.
4. Click the **Browse** button next to the **Project** field to select a project file to upload.
5. Click **Save**. The system adds the new project file to the database and displays the updated **Projects** page, if the operation is completed successfully.

To add multiple projects

1. Click the **Upload Project** button.
2. Click the Add files ... button and select your project files, or Select and drag and drop your files your files in the Drop files here area
3. Click **Save**. The system adds the new project file to the database and displays the updated **Projects** page, if the operation is completed successfully.

Editing a Project File

To edit a project file

1. On the Navigation Tree, click **Server Library** and select **Projects**.
2. Click the name of the project file you want to edit or click anywhere on its row, and then click .
3. Make the necessary changes. For details about how to change project parameters refer to [Adding a Project File](#) topic.
4. Click **Save**. The system saves the modified project file to the database and displays the updated Projects page, if the operation is completed successfully.

To assign project file to user groups, roles or users use the [Rights](#) tab.

To view, restore or edit project file revisions, use to the [Revisions](#) tab.

Assigning Project File Access Rights

The creator of a project file automatically receives the rights to view and edit it.

You can assign project file access rights to other users. Authorized users can view and edit project files, assigned to them, in MSAdmin.

To assign project file access rights

1. On the Navigation Tree, click **Server Library** and select **Projects**.
2. Click the name of a project file you want to assign access rights for.
3. On the displayed page, select the **Rights** tab.
4. Select **Global** check box for a model to be available to all users and click the **Save** button. Otherwise proceed to the next steps.
5. Click **User Groups** to expand a hidden section. Select user groups that you want to have access to the project file.
 - a. To add a user group drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a user group drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

6. Click **Roles** to expand a hidden section. Select roles that you want to have access to the project file.
 - a. To add a role drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a role drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

7. In the **Users** section select users that you want to have access to the project file.
 - a. To add a user group drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a user group drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

8. Click **Save** to apply the changes.

NOTE If you don't specify any access rights for a project file, then it will be available for all users.

Project File Revisions

You can view, edit, restore, or delete the necessary project file revision any time using the **Revisions** tab.

To edit a project file revision

1. On the Navigation Tree, click **Server Library** and select **Projects**.
2. Click the name of the project file you want to edit a revision for or click anywhere on its row, and then click .
3. On the displayed page, select the **Revisions** tab.
4. Click the Open icon  to open the project file in the Project Editor.
5. Edit the project file if necessary and click the **Commit** button. The Project page is displayed.
6. Click **Save**.
The system saves the modified project file as current and displays the updated Projects page, if the operation is completed successfully.

To restore a project file revision

1. On the Navigation Tree, click **Server Library** and select **Projects**.
2. Click the name of the project file you want to restore a revision for or click anywhere on its row, and then click .
3. On the displayed page, select the **Revisions** tab.
4. Choose the revision to be restored, and then click the corresponding Restore icon .
5. A prompt displays. Click **Ok** to proceed.
The restored revision occupies the latest entry in the list of revisions in the **Revisions** tab.

To delete a project file revision

1. On the Navigation Tree, click **Server Library** and select **Projects**.
2. Click the name of the project file you want to delete a revision for or click anywhere on its row, and then click .
3. On the displayed page, select the **Revisions** tab.
4. Choose which revision to delete by clicking anywhere on its row, and then click the **Delete** button  found at the end of the row or on the top left-hand side of the Revisions page.
5. A prompt displays. Click **Delete** to proceed with the deletion.

Opening a File in the Project Editor

To open a project file in the Project Editor

1. Click the  icon in the **Open** column. The Project Editor **General** tab is displayed.
2. Make necessary changes. For details refer to the "[Creating Project Files](#)" topic.
3. To save or download the modified project, click the corresponding button.

Deleting a Project File

To delete a project file from the list

1. On the Navigation Tree, click **Server Library** and select **Projects**.
2. Click  at the end of the row with the project file you want to delete. A dialog box appears.
3. Click **Delete** to delete the project file from the list.

NOTE

If a project file has the *Locked* status (), it cannot be deleted. [Unlock the project file](#) first and then try again.

If a project file is being used in a process, it cannot be deleted. Instead, a warning message pops up: "**There are objects that use this item: Processes: <Process Name>**."

Changing a Project File State

On the **Projects** page you can change model state to locked or unlocked. When a project file is locked, it cannot be edited or deleted.

To change project file state

1. On the Navigation Tree, click **Server Library** and select **Projects**.
2. Select a project file from the list and click the **Lock** icon at the end of the row:
 -  - means that a project file is locked
 -  - means that a project file is unlocked
3. To unlock all project files at once click the **Release all locks** icon ().

MODELS

A model file, created within Monarch, is a file with a .dmod or .xmod extension that may contain field, filter, summary and chart definitions as well as other settings. Typically, all Monarch models are created by a system administrator.

Creating a model file is one of the first steps you need to perform, as you cannot define a document type or archive documents into the system without specifying a model file. (Refer to the Altair documentation for detailed information on the model creation process.)

All models used in the Monarch Server system are listed on the Models page. Each model is defined by a Monarch model file.

To display the **Models** page, on the Navigation Tree, click **Server Library** and select **Models**.

You can use this page to do the following:

- [Add a model](#)
- [Edit a model](#)
- [Delete a model](#)

- [Change state of a model](#)

Adding a Model

To add a model

1. On the Navigation Tree, click **Server Library**, and select **Models**.
2. Click **+** on the **Models** page. The **Model** page appears.
3. Enter the name of the model file in the **Name** field.
4. Enter the description of the model in the **Description** field. This step is optional.
5. To enter the path to a model file in the **Model** field, click the **Browse** button, and select the model file.
6. Enter any comments if necessary in the **Comment** field.
7. Assign the model to a new or existing folder in the **Assign to Folders** field. To add the model to a new folder, click **Create New Folder**.
8. To save a model, click **Save**.
The system saves the new model to the database and displays the updated **Models** page, if the operation is completed successfully.
9. To reset the recent changes, click the **Clear** button.
10. To cancel the operation and return to the **Models** page, click the **Cancel** button.

To upload a model

1. On the Navigation Tree, click **Server Library** and select **Models**.
2. Click **Upload Models** on the **Models** page.
3. Click **Add Files** to browse to a model file or drag and drop model file in the **Drop files here** area.
4. To save a model, click **Save**.
The system saves the new model to the database and displays the updated **Models** page, if the operation is completed successfully.
5. To reset the recent changes, click the **Clear** button.
6. To cancel the operation and return to the **Models** page, click the **Cancel** button.

To replace model file paths

1. Expand the **Replace Paths** section. You will see a list of external models and lookup files. Those files that could not be found are marked with a warning sign.
2. Click the **Browse** button for the file, select the file path, then click **OK**.
3. Select the **Replace** checkbox for the file you want to replace path.
4. Click the **Save** button.

Editing a Model

Models can be edited via the Models page.

To edit a model

1. On the Navigation Tree, click **Server Library** and select **Models**.

2. Select the model you want to edit on the **Models** page:

Click the name of the model you want to edit or click anywhere on the row, and then click .

3. Make the necessary changes.

NOTE You can choose to check the modification applied to the model being edited. To do this, select the **Check dependencies on model fields (may slow down save operation)** checkbox.

For details about how to change model parameters refer to [Adding a Model](#) topic.

NOTE If changes are made to a model that is used by several projects, the new model (with changes) can be re-uploaded. On the **Models** page, click the **Browse** button, and select the new model. Proceed to Step 4. The new model is then used in the projects.

4. To save the changes, click the **Save** button.
The system saves the modified model to the database and displays the updated Models page, if the operation is completed successfully.

To view and restore model revisions

You can view or restore model revisions any time using the **Revisions** tab.

1. On the Navigation Tree, click **Server Library**, and select **Models**.
2. Click the name of the model file you want to restore a revision for or click anywhere on its row, and then click .
3. On the displayed page, select the **Revisions** tab.
4. Choose the revision to be restored, and then click the corresponding Restore icon .
5. A prompt displays. Click **Ok** to proceed.

The restored revision occupies the latest entry in the list of revisions in the **Revisions** tab.

To delete a model revision

1. On the Navigation Tree, click **Server Library** and select **Models**.
2. Click the name of the model file you want to delete a revision for or click anywhere on its row, and then click .
3. On the displayed page, select the **Revisions** tab.
4. Choose which revision to delete by clicking anywhere on its row, and then click the **Delete** button  found at the end of the row or on the top left-hand side of the Revisions page.
5. A prompt displays. Click **Delete** to proceed with the deletion.

To assign model to user groups, roles, or users

1. On the Navigation Tree, click **Server Library** and select **Models**.
2. Select the model you want to edit on the **Models** page. Click the name of the model you want to edit or click anywhere on the row, and then click .

3. Click the **Rights** tab. For more information, see [Assigning Model Access Rights](#).

Assigning Model Access Rights

The creator of a model automatically receives the rights to view and edit it.

You can assign model access rights to other users. Authorized users can view and edit models, assigned to them, in MSAdmin.

To assign model access rights

1. On the Navigation Tree, click **Server Library** and select **Models**.
2. Click the name of a model you want to assign access rights for.
3. On the displayed page, select the **Rights** tab.
4. Select **Global** check box for a model to be available to all users and click the **Save** button. Otherwise proceed to the next steps.
5. Click **User Groups** to expand a hidden section. Select user groups that you want to have access to the model.
 - a. To add a user group drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a user group drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

6. Click **Roles** to expand a hidden section. Select roles that you want to have access to the model.
 - a. To add a role drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a role drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

7. In the **Users** section select users that you want to have access to the model.
 - a. To add a user drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a user drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

8. To apply the changes, click the **Save** button.

NOTE If you don't specify any access rights for a model, then it will be available for all users.

Deleting a Model

You can delete a model from the system via the Models page.

NOTE

You cannot delete a model if it is currently assigned to a document type. You must first [remove it from the document type](#) in order to delete it from the system.

If you assign another model to a document type, you will need to [configure](#) and [run](#) a **Re-index maintenance task** to re-index the archived documents. If you do not re-index them, Monarch Server users will not be able to retrieve them.

To delete a model from the list

1. On the Navigation Tree, click **Server Library** and select **Models**.
2. Click  at the end of the row with the model you want to delete. A dialog box appears.
3. Click **Delete** to delete the model from the list.

NOTE

If a model has the *Locked* status (), it cannot be deleted. First [unlock the model](#) and then try again.

Changing Model State

On the **Models** page you can change model state to locked or unlocked. When a model is locked, it cannot be edited or deleted.

To change model state

1. On the Navigation Tree, click **Server Library** and select **Models**.
2. On the Models page, select a model from the list and click the **Lock** icon at the end of the row:
 -  - means that a model is locked
 -  - means that a model is unlocked
3. To unlock all models at once click the **Release all locks** icon () located below the model list.

TEMPLATES

Monarch Server users can apply templates to documents when exporting them. All templates used in the system are listed on the Templates page. To access this page, on the Navigation Tree, click **Server Library** and select **Templates**. You can use this page to do the following:

- [Add a template](#)
- [Edit a template](#)

- ❑ [Delete a template](#)
- ❑ [Change state of a template](#)

NOTE Each template type is available only under the appropriate license.

Adding a Template

To add a template

1. On the Navigation Tree, click **Server Library**, and select **Templates**.
2. On the displayed page, click the **Add** icon .
3. Enter the name of a new template in the **Name** field.
4. Enter template description if necessary in the **Description** field.
5. Select template type from the drop-down list in the **Type** field.
6. Click the **Browse** button to locate the template file.
7. Enter any comments if necessary in the **Comment** field.
8. Assign the template to a new or existing folder in the **Assign to Folders** field. To add the template to a new folder, click **Create New Folder**.
9. To save changes, click **Save**.
The system saves the new template to the database and displays the updated Templates page, if the operation is completed successfully.
10. To reset the recent changes, click the **Clear** button.
11. To cancel the operation and return to the **Templates** page, click the **Cancel** button.

To upload a template

1. On the Navigation Tree, click **Server Library** and select **Templates**.
2. Click Upload Templates on the Templates page.
3. Click **Add Files** to browse to a template file or drag and drop template file in the **Drop files here** area.
4. To save a template, click **Save**.
The system saves the new template to the database and displays the updated **Templates** page, if the operation is completed successfully.
5. To reset the recent changes, click the **Clear** button.
6. To cancel the operation and return to the **Templates** page, click the **Cancel** button.

Editing a Template

To edit a template

1. On the Navigation Tree, click **Server Library** and select **Templates**.
2. Click the name of the template you want to edit on the **Templates** page or click anywhere on the row, and then click .

3. Make the necessary changes. For details about how to change template parameters refer to [Adding a Template](#) topic.
4. Click the **Rights tab** to assign template to user groups, roles or users. They can then apply the templates to exported documents.
5. Click **Save**. The system saves the modified template to the database and displays the updated Templates page, if the operation is completed successfully.

To view and restore template revisions

You can view or restore template revisions any time using the **Revisions** tab.

1. On the Navigation Tree, click **Server Library**, and select **Templates**.
2. Click the name of the template file you want to restore a revision for or click anywhere on its row, and then click .
3. On the displayed page, select the **Revisions** tab.
4. Choose the revision to be restored, and then click the corresponding Restore icon .
5. A prompt displays. Click **Ok** to proceed.
The restored revision occupies the latest entry in the list of revisions in the **Revisions** tab.

To delete a template revision

1. On the Navigation Tree, click **Server Library** and select **Templates**.
2. Click the name of the template file you want to delete a revision for or click anywhere on its row, and then click .
3. On the displayed page, select the **Revisions** tab.
4. Choose which revision to delete by clicking anywhere on its row, and then click the **Delete** button  found at the end of the row or on the top left-hand side of the Revisions page.
5. A prompt displays. Click **Delete** to proceed with the deletion.

To assign template to user groups, roles, or users

1. On the Navigation Tree, click **Server Library** and select **Templates**.
2. Select the template you want to edit on the **Templates** page. Click the name of the template you want to edit or click anywhere on the row, and then click .
3. Click the **Rights** tab. For more information, see [Assigning Template Access Rights](#).

Assigning Template Access Rights

The creator of a template automatically receives the rights to view and edit it.

You can assign template access rights to other users. Authorized users can view, edit and use templates assigned to them in MSAdmin.

To assign template access rights

1. On the Navigation Tree, click **Server Library** and select **Templates**.
2. Click the name of a template you want to assign access rights for.
3. On the displayed page, select the **Rights** tab.
4. Select **Global** check box for a template to be available to all users and click the **Save** button. Otherwise proceed to the next steps.

5. Click **User Groups** to expand a hidden section. Select user groups that you want to have access to the template.
 - a. To add a user group drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a user group drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

6. Click **Roles** to expand a hidden section. Select roles that you want to have access to the template.
 - a. To add a role drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a role drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

7. In the **Users** section you can select users that you want to have access to the template.
 - a. To add a user drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a user drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

8. To apply the changes, click the **Save** button.

NOTE If you don't specify any access rights for a template file, then it will be available for all users.

Deleting a Template

You can delete a template from the system via the Templates page.

NOTE You cannot delete a template if it is currently assigned to a document type. You must first [remove it from the document type](#) in order to delete it from the system.

To delete a template from the list

1. On the Navigation Tree, click **Server Library** and select **Templates**.
2. Click  at the end of the row with the template you want to delete. A dialog box appears.

3. Click **Delete** to delete the template from the list.

NOTE

If a template has the *Locked* status (), it cannot be deleted. First [unlock the template](#) and then try again.

Changing Template State

On the **Templates** page you can change template state to locked or unlocked. When a template is locked, it cannot be edited or deleted.

To change template state

1. On the Navigation Tree, click **Server Library** and select **Templates**.
2. On the Templates page, select a template from the list and click the **Lock** icon at the end of the row:
 -  - means that a template is locked
 -  - means that a template is unlocked
3. To unlock all templates at once click the **Release all locks** icon () located below the template list.

WORKSPACES

Workspace is the collective term for the objects and actions that are involved in a data preparation operation. Your workspace includes:

- the tables you have loaded;
- the join actions you have defined and the resulting tables that are produced;
- the prep data operations you have carried out.

To display the **Workspaces** page, on the Navigation Tree, click **Server Library** and select **Workspaces**.

You can use this page to do the following:

- [Add a workspace](#)
- [Edit a workspace](#)
- [Delete a workspace](#)
- [Change state of a workspace](#)

Adding a Workspace

To add a workspace

1. On the Navigation Tree, click **Server Library**, and select **Workspaces**.
2. Click  on the **Workspaces** page. The **Workspaces** page appears.
3. Enter the name of a workspace file in the **Name** field.

4. Enter the description of a workspace in the **Description** field. This step is optional.
5. To enter the path to a workspace file in the **Workspace** field, click the **Browse** button, and select a workspace file.

NOTE Workspace files that are protected by passwords can also be uploaded to the Server Library. When a password-protected file is selected, a dialog box asking for a password appears. Enter the password in the dialog box to add the desired workspace.

In addition to Excel file types .xls and .xlsx, .xlsb and .xlsm file types may be used as database file input.

6. Enter any comments if necessary in the **Comment** field.
7. Assign the workspace to a new or existing folder in the **Assign to Folders** field. To add the workspace to a new folder, click **Create New Folder**.
8. To save a workspace, click **Save**.
The system saves the new workspace to the database and displays the updated **Workspaces** page, if the operation is completed successfully.
9. To reset the recent changes, click the **Clear** button.
10. To cancel the operation and return to the **Workspaces** page, click the **Cancel** button.

To upload a workspace

1. On the Navigation Tree, click **Server Library** and select **Workspaces**.
2. Click **Upload Workspaces** on the Workspaces page.
3. Click **Add Files** to browse to a workspace file or drag and drop workspace file in the **Drop files here** area.
4. To save a workspace, click **Save**.
The system saves the new workspace to the database and displays the updated **Workspaces** page, if the operation is completed successfully.
5. To reset the recent changes, click the **Clear** button.
6. To cancel the operation and return to the **Workspaces** page, click the **Cancel** button.

To replace workspace file paths

1. Expand the **Replace Paths** section. You will see a list of workspace input files, along with the names of the load plans. Those files that could not be found are marked with a warning sign.
2. Click the **Browse** button for the file, select the file path, then click **OK**. The file can also be dragged and dropped into the rectangular area to overwrite the input file.

When the workspace being added is based on a database (e.g., SQL Server, OLEDB, ODBC, Google Analytics, or Oracle), the connector settings for the workspace can be modified as follows:

- a. In the expanded **Replace Paths** section, click **Edit Table Info**.
- b. An interface appears. Edit the necessary parameters.
 - ◆ For a workspace based on SQL server, e.g., Host Name, Port, Authentication, User ID, Password. (**NOTE:** Click **Validate** to test the connection based on the SQL plugin settings provided)
 - ◆ For a workspace based on OLEDB, e.g., Connection Name, SQL Dialect, Timeout.
 - ◆ For a workspace based on ODBC, e.g., ODBC DSN, User ID, and Password.

- ◆ For a workspace based on Google Analytics, e.g., Service Account Email, Profile ID, Start Date, End Date.
 - ◆ For a workspace based on Oracle, e.g., Connection Type, Host Name, Port, Service Name, User ID, Password, Other Connection Options, Edition Name, Tables and Views, Query, etc.
- c. Click **Ok**.

NOTE When the workspace being added contains Excel load plans (but not Excel Trapping load plans), the Excel Engine (i.e., Legacy or Modern) to use can be selected from the drop-down menu in the **Additional Settings** column.

The **Replace** checkbox corresponding to the file whose path was updated is automatically selected.

3. Click the **Save** button.

NOTE After you upload new file for input that is linked with the current workspace, it is marked as pinned. Pinned files are displayed for each workspace. If several load plans within a workspace contain an unpinned file, it will appear in the list only once. All pinned files are displayed separately.

Editing a Workspace

Workspaces can be edited via the **Workspaces** page.

To edit a workspace

1. On the Navigation Tree, click **Server Library** and select **Workspaces**.
2. Select the workspace you want to edit on the **Workspaces** page:
3. Click the name of the workspace you want to edit or click anywhere on the row, and then click .
4. Make the necessary changes. For details about how to change workspace parameters refer to [Adding a Workspace](#) topic.
5. To the changes, click the **Save** button.
The system saves the modified workspace to the database and displays the updated Workspaces page, if the operation is completed successfully.

To view and restore workspace revisions

You can view or restore workspace revisions any time using the **Revisions** tab.

1. On the Navigation Tree, click **Server Library**, and select **Workspaces**.
2. Click the name of the workspace file you want to restore a revision for or click anywhere on its row, and then click .
3. On the displayed page, select the **Revisions** tab.
4. Choose the revision to be restored, and then click the corresponding Restore icon .
5. A prompt displays. Click **Ok** to proceed.

The restored revision occupies the latest entry in the list of revisions in the **Revisions** tab.

To delete a workspace revision

1. On the Navigation Tree, click **Server Library** and select **Workspaces**.
2. Click the name of the workspace file you want to delete a revision for or click anywhere on its row, and then click .
3. On the displayed page, select the **Revisions** tab.
4. Choose which revision to delete by clicking anywhere on its row, and then click the **Delete** button  found at the end of the row or on the top left-hand side of the Revisions page.
5. A prompt displays. Click **Delete** to proceed with the deletion.

To assign workspace to user groups, roles, or users

1. On the Navigation Tree, click **Server Library** and select **Workspaces**.
2. Select the model you want to edit on the **Workspaces** page. Click the name of the workspace you want to edit or click anywhere on the row, and then click .
3. Click the Rights tab. For more information, see [Assigning Workspace Access Rights](#).

Assigning Workspace Access Rights

The creator of a workspace automatically receives the rights to view and edit it.

You can assign workspace access rights to other users. Authorized users can view and edit workspaces, assigned to them, in MSAdmin.

To assign workspace access rights

1. On the Navigation Tree, click **Server Library** and select **Workspaces**.
2. Click the name of a workspace you want to assign access rights for.
3. On the displayed page, select the **Rights** tab.
4. Select **Global** check box for a workspace to be available to all users and click the **Save** button. Otherwise proceed to the next steps.
5. Click **User Groups** to expand a hidden section. Select user groups that you want to have access to the workspace.
 - a. To add a user group drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a user group drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

6. Click **Roles** to expand a hidden section. Select roles that you want to have access to the workspace.
 - a. To add a role drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a role drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

7. In the **Users** section select users that you want to have access to the workspace.
 - a. To add a user drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a user drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

8. To apply the changes, click the **Save** button.

NOTE If you don't specify any access rights for a workspace, then it will be available for all users.

Deleting a Workspace

You can delete a workspace from the system via the Workspaces page.

To delete a workspace from the list

1. On the Navigation Tree, click **Server Library** and select **Workspaces**.
2. Click  at the end of the row with the workspace you want to delete. A dialog box appears.
3. Click **Delete** to delete the workspace from the list.

NOTE If a workspace has the *Locked* status (), it cannot be deleted. First [unlock the workspace](#) and then try again.

Changing Workspace State

On the **Workspaces** page you can change workspace state to locked or unlocked. When a workspace is locked, it cannot be edited or deleted.

To change workspace state

1. On the Navigation Tree, click **Server Library** and select **Workspaces**.
2. On the Workspace page, select a workspace from the list and click the **Lock** icon at the end of the row:
 -  - means that a workspace is locked
 -  - means that a workspace is unlocked

3. To unlock all workspaces at once click the **Release all locks** icon () located below the workspace list.

CHANGE LISTS

Monarch Personal tracks all transformations that you carry out on a table and logs these in a change history list. Note that changes are only tracked in the Prep Data Window.

Using Change History, you can:

- have a ready audit trail so you can review the transformations you have carried out;
- undo changes;
- apply changes to other columns in other tables within the Prep Data Window. You can even export the Change History and apply to other tables in a different workspace.
- create a library of changes that you can import and reuse.

To display the **Change Lists** page, on the Navigation Tree, click **Server Library** and select **Change Lists**.

You can use this page to do the following:

- [Add a change list](#)
- [Edit a change list](#)
- [Delete a change list](#)
- [Change state of a change list](#)

Adding a Change List

To add a change list

1. On the Navigation Tree, click **Server Library**, and select **Change Lists**.
2. Click  on the **Change Lists** page. The **Change Lists** page appears.
3. Enter the name of a change list file in the **Name** field.
4. Enter the description of a change list in the **Description** field. This step is optional.
5. To enter the path to a change list file in the **Change List** field, click the **Browse** button, and select a change list file.
6. Enter any comments if necessary in the **Comment** field.
7. Assign the change list to a new or existing folder in the **Assign to Folders** field. To add the change list to a new folder, click **Create New Folder**.
8. To save a change list, click **Save**.
The system saves the new change list to the database and displays the updated **Change Lists** page, if the operation is completed successfully.
9. To reset the recent changes, click the **Clear** button.
10. To cancel the operation and return to the **Change Lists** page, click the **Cancel** button.

To upload a change list

1. On the Navigation Tree, click **Server Library** and select **Change Lists**.
2. Click **Upload Change Lists** on the Change Lists page.

3. Click **Add Files** to browse to a change list file or drag and drop change list file in the **Drop files here** area.
4. To save a change list, click **Save**.
The system saves the new change list to the database and displays the updated **Change Lists** page, if the operation is completed successfully.
5. To reset the recent changes, click the **Clear** button.
6. To cancel the operation and return to the **Change Lists** page, click the **Cancel** button.

Editing a Change List

Change Lists can be edited via the Change Lists page.

To edit a change list

1. On the Navigation Tree, click **Server Library** and select **Change Lists**.
2. Select the Change List you want to edit on the **Change Lists** page:

Click the name of the change list you want to edit or click anywhere on the row, and then click .
3. Make the necessary changes. For details about how to change list parameters refer to [Adding a Change List](#) topic.
4. To the changes, click the **Save** button.
The system saves the modified change list to the database and displays the updated Change Lists page, if the operation is completed successfully.

To view and restore change list revisions

You can view or restore change list revisions any time using the **Revisions** tab.

1. On the Navigation Tree, click **Server Library**, and select **Change Lists**.
2. Click the name of the change list file you want to restore a revision for or click anywhere on its row, and then click .
3. On the displayed page, select the **Revisions** tab.
4. Choose the revision to be restored, and then click the corresponding Restore icon .
5. A prompt displays. Click **OK** to proceed.
The restored revision occupies the latest entry in the list of revisions in the **Revisions** tab.

To delete a change list revision

1. On the Navigation Tree, click **Server Library** and select **Change Lists**.
2. Click the name of the change list file you want to delete a revision for or click anywhere on its row, and then click .
3. On the displayed page, select the **Revisions** tab.
4. Choose which revision to delete by clicking anywhere on its row, and then click the **Delete** button  found at the end of the row or on the top left-hand side of the Revisions page.
5. A prompt displays. Click **Delete** to proceed with the deletion.

To assign change list to user groups, roles, or users

1. On the Navigation Tree, click **Server Library** and select **Change Lists**.
2. Select the model you want to edit on the **Change Lists** page. Click the name of the change list you want to edit or click anywhere on the row, and then click .

3. Click the **Rights** tab. For more information, see [Assigning Change List Access Rights](#).

Assigning Change List Access Rights

The creator of a change list automatically receives the rights to view and edit it.

You can assign change list access rights to other users. Authorized users can view and edit change lists, assigned to them, in MSAdmin.

To assign change list access rights

1. On the Navigation Tree, click **Server Library** and select **Change Lists**.
2. Click the name of a change list you want to assign access rights for.
3. On the displayed page, select the **Rights** tab.
4. Select **Global** check box for a change list to be available to all users and click the **Save** button. Otherwise proceed to the next steps.
5. Click **User Groups** to expand a hidden section. Select user groups that you want to have access to the change list.
 - a. To add a user group drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a user group drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

6. Click **Roles** to expand a hidden section. Select roles that you want to have access to the change list.
 - a. To add a role drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a role drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

7. In the **Users** section select users that you want to have access to the change list.
 - a. To add a user drag it from the left pane to the right pane or click  icon next to its name
 - b. To remove a user drag it from the right pane to the left pane or click  icon next to its name.

NOTE You may also use **Add All** and **Remove All** controls.

8. To apply the changes, click the **Save** button.

NOTE If you don't specify any access rights for a workspace, then it will be available for all

users.

Deleting a Change List

You can delete a change list from the system via the Change Lists page.

To delete a change list from the list

1. On the Navigation Tree, click **Server Library** and select **Change List**.
2. Click  at the end of the row with the change list you want to delete. A dialog box appears.
3. Click **Delete** to delete the change list from the list.

NOTE

If a change list has the *Locked* status (), it cannot be deleted. First [unlock the change list](#) and then try again.

Changing Change List State

On the **Change Lists** page you can change a change list state to locked or unlocked. When a change list is locked, it cannot be edited or deleted.

To change a change list state

1. On the Navigation Tree, click **Server Library** and select **Change Lists**.
2. On the Change Lists page, select a change list from the list and click the **Lock** icon at the end of the row:
 -  - means that a change list is locked
 -  - means that a change list is unlocked
3. To unlock all change lists at once click the **Release all locks** icon () located below the change list.

BULK EDITS

The **Bulk Edits** feature of the Monarch Server administrator allows you to apply text content bulk editing to models, alerts, distributions, and project objects. Input and manifest files within monitored Standard and Visual Processes can also be bulk edited.

To use bulk editing

1. On the Navigation Tree, click **Server Library**, and select **Bulk Edits**.
2. On the displayed page, enter the text fragment you want to edit in the **Find** text box. You can combine your text fragments with "*" and "?" wildcards in the Find text box to leverage your search query. If you select the **Ignore Wildcards** check box, the wildcards will not be applied during the search.
3. Enter the text fragment with which you want to replace the original fragment in the **Replace** text box.

4. Define the appropriate Server Library (**Alerts** [global alerts for standard processes], **Distributions** [global distributions for standard processes], **Models**, **Projects**, **Templates**, **Workspaces**, **Change lists**, **Visual Processes**, and **Standard Processes**) where you want to perform bulk editing by selecting the corresponding check boxes in the **Find in** line. You can assign bulk editing to the whole definition object or to its particular settings, for example, to Channel links and titles of the alerts log only. Consider the following options:
 - In the **Find in** line, select the check boxes corresponding to the objects to which bulk editing will be applied.
 - If you want to apply bulk editing to all models, for example, select the Models checkbox.
 - If you want to apply bulk editing to the descriptions of models only, click Models to display the context menu. From the menu that displays, select the Description check box.

NOTE

Some of the options available for bulk editing are as follows:

- You can apply bulk editing to all Server Library by selecting all check boxes.
- To replace ACE DAO/ACE OleDb/Default values for Visual Processes, bulk editing can be applied to **Export behavior for accdb/dbf/mdb**. Under Visual Processes, select Table or Summary Export, and then choose **Export behavior for accdb/dbf/mdb**.
- The following properties of distribution items in Visual Processes can also be modified via Bulk Edits: Copy, Move, Rss, Email, Email by key, Log, and Export to Amazon S3.
- Scripts for Standard and Visual Processes can be edited via Bulk Edits. The following options are available:

For Standard Processes:

- Script namespaces
- Script references
- Script global declaration
- Script pre export
- Script post export

For Visual Processes:

- Script namespaces
- Script references
- Script global declaration

5. Click **Find** if you want just view the results of your search query. The results are displayed in the Search Results table that contains the detailed definition object information, namely, the object name, its settings and values. In the table, you can mark out all or some required object items for further replacement.
6. Click **Replace** if you want to apply the assigned bulk editing for the specified Server Library. Executed changes are irreversible.

NOTE

When searching values using a wildcard (*) and then proceeding with the replace operation without specifying a new search criterion, the following alert displays:
"Warning: You are about to use the "Replace" option using a "wildcard" search

pattern. Doing so will replace different types with the replace value specified. Do you want to continue?" Click **OK** to proceed with the replace operation; alternatively, click **Cancel**.

RIGHTS AND PRIVILEGES

Monarch Server users should be given privileges to carry out such tasks as defining searches, creating notifications, etc.

The Monarch Server administrator should also provide users with the rights to access objects.

In the following sections you will learn:

- ❑ [How to assign privileges to roles, user groups and users](#)
- ❑ [How to assign access rights](#)
- ❑ [How to set a security provider](#)
- ❑ [How to use security keywords](#)
- ❑ [How to manage roles](#)
- ❑ [How to manage user groups](#)
- ❑ [How to manage users](#)
- ❑ [How to manage document state groups](#)

ASSIGNING PRIVILEGES

When you grant privileges to [roles](#), [user groups](#) or [users](#), you specify which actions the role members, group members or users can perform within Monarch Server.

The following privileges can be granted:

- ❑ Common privileges
 - **Administrator:** Allows users to log on to MSAdmin.
 - **Portlets:** Allows users to view portlets — Panopticon Workbooks — in Client.
- ❑ Automator privileges
 - **Automator Administrator:** Allows the user full administration.
 - **Automator User:** Allows the user to manage processes and jobs.
 - ◆ **Can Edit Scripts:** Allows the user to edit process scripts and script items for Visual Processes and Standard Processes, as well as project scripts (i.e., PreProcess, PostProcess, and Global). If unselected, the user cannot edit the properties of script items for existing processes; the user can only run the script and check the results.
 - **Automator Light User:** Allows the user to partially manage processes and jobs.
 - ◆ **Can Edit Schedules:** Allows the user to add, edit, and delete schedules for Visual Processes and Standard Processes. Note that only the schedules whose processes are assigned to the user can be modified.
- ❑ RMS privileges
 - **RMS User:** Allows the user to connect to RMS Client.
 - **RMS Administrator:** Provides the user with full administration rights to Monarch Server Report Mining Edition.

To grant privileges

1. On the Navigation Tree, click **Rights and Privileges**, and select **Roles**, **User Groups** or **Users** (depending on who you want to assign privileges to).
2. Click the name of a role, user group, or user by clicking anywhere on the row, and then click .
3. In the **Privileges** section, select the necessary check boxes to specify the privileges for the role, user group, or user.
4. Click **Save** to save the current settings.

Automator Administrator

The Automator Administrator user has the right for full administration of all the Automator components.

Automator User

The Automator User has the right to manage the following components.

- Automator
 - Standard Processes
 - ◆ **Processes:** The user can add a new Standard Process and edit, delete, and run Standard Processes that are assigned to him.
 - ◆ **Schedules:** The user can edit, delete, and run schedules whose Standard Processes are assigned to him.
 - ◆ **Jobs:** The user can edit, delete, and run the jobs whose Standard Processes are assigned to him.
 - ◆ **Project Editor:** The user can create project files and edit, lock, or delete the project files assigned to him.
 - Visual Processes
 - ◆ **Processes:** The user can add a new Visual Process and edit, delete, and run Visual Processes that are assigned to him.
 - ◆ **Schedules:** The user can edit, delete, and run schedules whose Visual Processes are assigned to him.
 - ◆ **Templates:** The user can add, edit, and delete templates assigned to him.
 - ◆ **Logs:** The user can edit, delete, and run the logs whose Visual Processes are assigned to him.
 - **Connection Definition:** The user can add, edit, and delete connection definitions.
- Server Library:** The user can add folders, projects, models, workspaces, and change lists to the system and edit, lock, or delete those that are assigned to him. The user can also perform bulk edits on the text content of models, projects, workspaces, change lists, Visual Processes, and Standard Processes that are available to him.

Automator Light User

The Automator Light User has the right to manage the following components:

- Standard Processes**
 - **Processes:** The user can view and run the Standard Processes that are assigned to him.
 - **Schedules:** The user can add, edit, and delete schedules whose Standard Processes are assigned to him.
 - **Jobs:** The user can view, run, delete, and cancel the jobs whose Standard Processes are assigned to him.
- Visual Processes**
 - **Processes:** The user can view and run the Visual Processes that are assigned to him.

- **Schedules:** The user can add, edit, and delete schedules whose Visual Processes are assigned to him.
- **Logs:** The user can view, run, delete, and cancel the logs whose Visual Processes are assigned to him.

NOTE Schedules are only available when the Automator Light User is granted the “Can Edit Schedules” privilege.

ASSIGNING ACCESS RIGHTS

When you assign access rights to a [role](#), [user group](#) or [user](#), you grant them access to a particular part of the MS system. For example, if you assign document type rights to a user group, all members of that user group will be able to retrieve documents of the specified document type.

NOTE Keep in mind that access rights in MS are cumulative (for example, access rights of a role are added to those of any user or user group assigned to it).

To assign access rights

1. On the Navigation Tree, click **Rights and Privileges**, and select **Roles**, **User Groups** or **Users**, depending on who you want to assign rights to.
2. Select a role, user group, or a user. Click its name or click anywhere on the row, and then click .
3. Select a tab to specify appropriate rights at the top of the page. The following tabs are available:
 - **Views:** Allows to grant access to [views](#) for the role, user group or user in the document search results page or RMS views.
 - **Models** tab allows you to assign [models](#) to the current user.
 - **Templates:** Allows to grant access to [templates](#) templates for the role, user group or user.
 - **Portlets:** Allows to grant access to specific portlets for the role, user group or user.
 - **Automator Processes:** Allows to grant access to automator processes for the role, user group or user.
 - **Visual Processes:** Allows to grant access to visual process for the role, user group or user.
 - **Automator Projects:** Allows to grant access to projects for the role, user group or users.
 - **Workspaces:** Allows to grant access to workspaces for the role, user group or users.
 - **Change Lists:** Allows to grant access to change lists for the role, user group or users.
4. Assign the appropriate rights.
5. To save changes, click **Save**.
6. Repeat steps 3-5 until you have assigned all desired rights to a role, user group or user.

SETTING SECURITY PROVIDER

A security provider provides the system with security information about user groups and users, as well as checks user passwords for validity.

To select the security provider of the system, on the Navigation Tree, click **Rights and Privileges**, and select **Security Provider**.

You can select one of the following options:

- Active Directory**. The system is supported by Active Directory Authentication, which is based on (and integrated with) the security system of Windows NT.
- Default**. The default provider presents an ordinary security model dealing with roles, groups and users, which are kept in the MS database. The default provider does not interact with security systems.
- Strong password provider**. The Strong Password Provider supports the same security model as the default provider, but applies stricter rules to passwords. In particular it supports password history, length limitation and expiration period. If you select this option, you can specify the following settings:
 - **Number of passwords stored in the history**: The valid values are integers in the range 0 - 100. This value specifies the number of last used passwords that the system remembers for each user. The system does not allow assigning a new password, if it is in the password history. Zero value disables this functionality. The default value is 15.
 - **Minimum password length**: The valid values are integers in the range 0 - 128. The value specifies the minimum password length. Zero value disables this functionality. The default value is equal to 7.
 - **Password age in days**: The valid values are integers in the range 0 - 365. The value specifies the number of days after which the password will expire. Zero value disables this functionality. The default value is 30 days.
 - **Password must meet complexity requirements**: This check box enables validation of complexity requirements for the password. These requirements include the following rules: within the password string user must use at least three characters out of the four types of characters: digits (0-9), upper case alpha characters (A-Z), lower case alpha characters (a-z), and special characters which are pretty much the rest of the keyboard.
 - **Force a new user to change the password at first logon**: If you select this check box, a user must change the password at the first logon, in case an administrator does not set a password for the user.

NOTE All the settings of the Strong Password Provider are global and have effect for all users.

- Show SSO**: Select this check box and the **Active Directory** option, to enable the user to sign in to the system by clicking the **SSO** link on the Login page, without having to type the name and password each time.
- Enable Active Directory Group Lookup**: to allow searching and importing user groups on the [Importing a User Group](#) page.

ROLES

Roles are used to set rights and privileges for users and user group. A role can include both [user groups](#) and [users](#), but cannot include other roles. A role is kept in the database and has no tie with other security systems. An administrator can define rights via roles (Role Based Security) or assign them to user groups and users individually.

All roles used in the system are listed on the Roles page. To display the Roles page, on the Navigation Tree, click **Rights and Privileges** and select **Roles**.

Use this page to set rights and privileges for roles, as well as to [add](#), [edit](#), or [remove](#) roles from the list.

Adding a Role

Roles allow you to quickly assign specific [access rights](#) and [privileges](#) to a [user](#) or [user group](#). Because access rights and privileges in Monarch Server are cumulative, when you add a user or user group to a role, the user or user group members are granted the role's access rights and privileges in addition to their own.

To add a role

1. On the Navigation Tree, click **Rights and Privileges**, and select **Roles**.
2. Click  on the **Roles** page. The **Role** page containing the following items appears:
 - **Role Name:** Enter a unique name of a new role.
 - **Description:** Optionally, enter any text defining the role.
 - **Privileges:** By selecting check boxes, an administrator can grant the following privileges:
 - ◆ **Administrator:** Allows the user to log on to MSAdmin.
 - ◆ **Portlets:** Allows the user to view Panopticon Workbooks in MSCClient.
 - ◆ **Automator Administrator:** Allows the user full administration.
 - ◆ **Automator User:** Allows the user to manage processes and jobs.
 - **Can Edit Scripts:** Allows the user to edit process scripts and script items for Visual Processes and Standard Processes, as well as project scripts (i.e., PreProcess, PostProcess, and Global). If unselected, the user cannot edit the properties of script items for existing processes; the user can only run the script and check the results.
 - ◆ **Automator Light User:** Allows the user to partially manage processes and jobs.
 - **Can Edit Schedules:** Allows the user to add, edit, and delete schedules for Visual Processes and Standard Processes. Note that only the schedules whose processes are assigned to the user can be modified.
 - ◆ **RMS User:** Allows the user to connect to RMS Client.
 - ◆ **RMS Administrator:** Allows the user to log on to RMSAdmin.
 - **Members:** Click this heading to expand hidden sections.
 - ◆ **Available User Groups:** Displays all available user groups from which you can select ones to assign to the current role. To add a user group, drag it from the left pane to the right pane.
 - ◆ **Selected User Groups:** The user groups in this pane will be assigned to the current role. To remove a user group, drag it from the right pane to the left pane.
 - ◆ **Available Users:** Displays all available users from which you can select ones to assign to the current role. To add a user, drag it from the left pane to the right pane.
 - ◆ **Selected Users:** The users in this pane will be assigned to the current role. To remove a user, drag it from the right pane to the left pane.
 - **Save:** Click to save a new role and close the page.
 - **Clear:** Click to clear all the boxes.
 - **Save/Edit:** Click to save a new role and display additional tabs.
 - **Cancel:** Click to cancel the operation.
 - **Copy Profile:** Opens a dialog box, where you can select a role. All its settings will be copied. You can make the necessary changes.
3. Fill in the boxes of the page and click **Save**. The system saves the new role to the database and displays the updated Roles page, if the operation is completed successfully.

NOTE After you have created a role, you can assign access rights to it. For details, see [Assigning Access Rights](#).

Editing a Role

To edit a role

1. On the Navigation Tree, click **Rights and Privileges**, and select **Roles**.
2. Click the name of the role you want to edit or click anywhere on the row, and then click . The **Role** page appears.
3. Make the necessary changes. For more information on how to change its items, see [Adding a Role](#).
4. To grant access rights, select an appropriate tab at the top of the page. The following tabs are available in the **Edit** mode:
 - **Role tab** allows you to modify the role name and description and grant common and specific privileges to the selected role.
 - **Views tab** allows you to specify what [views](#) of the document search results page or RMS views page will be available for the current role. To do this, select or clear appropriate check boxes, and click **Save**.
 - **Settings tab** allows you to modify the [notification settings](#), particularly the number of days when an alert about license expiration will be shown.

NOTE The **Defined** check box is disabled by default. If the setting is not “Defined,” then the global setting under System Preferences applies.

- **Portlets tab** allows you to [assign portlet access rights](#) to the current user role.
- **Templates tab** allows you to assign document [templates](#) to the current role. To do this, select a document type from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.
- **Models tab** allows you to assign [models](#) to the current user. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
- **Automator Processes tab** allows you to assign Automator processes to the current role. To do this, select a search from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.
- **Visual Processes tab** allows you to assign visual processes to the current role. To do this, select a search from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.

NOTE Processes created in older versions of Monarch Server (e.g., v13.2) are assigned to all users.

- **Automator Projects tab** allows you to assign projects to the current role. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
- **Workspaces tab** allows you to assign workspaces to the current role. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
- **Change Lists tab** allows you to assign change lists to the current role. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.

- **Folders** tab allows you to assign folders to the current role. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
 - **Named Paths** tab allows you to assign named paths to the current role. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
5. Make the necessary changes and click **Save**.
The system saves the modified role to the database and displays the updated **Roles** page, if the operation is completed successfully.

Deleting a Role

To delete a role

1. On the Navigation Tree, click **Rights and Privileges**, and select **Roles**.
2. Click **X** at the end of the row with the role you want to delete. A dialog box appears.
3. Click **Delete** to delete the role from the list.

USER GROUPS

User groups allow you to quickly assign specific [access rights](#) and [privileges](#) to [users](#).

A user group can include both user groups and users. A user group is kept in the database and has no tie with other security systems. An administrator can define rights via user groups or assign them to users individually.

Access rights and privileges in Monarch Server are cumulative. It means that when you add a user to a user group, the user is granted the group's access rights and privileges.

All user groups used in the system are listed on the User Groups page. To display the User Groups page, on the Navigation Tree, click **Rights and Privileges** and select **User Groups**.

Use this page to [set access rights](#) and [privileges](#) for user groups, as well as to [add](#), [import](#), [edit](#), or [remove](#) user groups from the list.

Adding a User Group

To add a user group

1. On the Navigation Tree, click **Rights and Privileges**, and select **User Groups**.
2. Click **+** on the **User Groups** page. The **User Group** page containing the following items appears:
 - **User Group Name:** Enter a unique name of a new user group.
 - **Description:** Optionally, enter any text defining the user group.
 - **Alias:** Active Directory group name. This field is filled out automatically if you apply user group importing.

NOTE **Alias** is only available when the **Enable Active Directory Group Lookup** checkbox in **Active Directory** is selected.

- **Privileges:** By selecting check boxes, an administrator can grant the following privileges:
 - ◆ **Administrator:** Allows the user to log on to MSAdmin.

- ◆ **Portlets:** Allows the user to view Panopticon Workbooks in MSCClient.
 - ◆ **Automator Administrator:** Allows the user full administration.
 - ◆ **Automator User:** Allows the user to manage processes and jobs.
 - **Can Edit Scripts:** Allows the user to edit process scripts and script items for Visual Processes and Standard Processes, as well as project scripts (i.e., PreProcess, PostProcess, and Global). If unselected, the user cannot edit the properties of script items for existing processes; the user can only run the script and check the results.
 - ◆ **Automator Light User:** Allows the user to partially manage processes and jobs.
 - **Can Edit Schedules:** Allows the user to add, edit, and delete schedules for Visual Processes and Standard Processes. Note that only the schedules whose processes are assigned to the user can be modified.
 - ◆ **RMS User:** Allows the user to connect to RMS Client.
 - ◆ **RMS Administrator:** Allows the user to log on to RMSAdmin.
 - **Members:** Click this heading to expand the hidden sections.
 - ◆ **Available User Groups:** Displays all available user groups from which you can select ones to assign to the current user group. To select a user group, drag it from the left pane to the right pane.
 - ◆ **Selected User Groups:** The user groups in this pane will be assigned to the current user group. To remove a user group, drag it from the right pane to the left pane.
 - ◆ **Available Users:** Displays all available users from which you can select ones to assign to the current user group. To add a user, drag it from the left pane to the right pane.
 - ◆ **Selected Users:** The users in this pane will be assigned to the current user group. To remove a user, drag it from the right pane to the left pane.
 - **Members of:** Click this heading to expand the hidden sections.
 - ◆ **Available User Groups:** Displays all available user groups from which you can select ones to assign the current user group to. To add a user group, drag it from the left pane to the right pane.
 - ◆ **Selected User Groups:** The current user group will be assigned to the user groups in this pane. To remove a user group, drag it from the right pane to the left pane.
 - ◆ **Available User Roles:** Displays all available user roles from which you can select ones to assign to the current user group. To select a user role, drag it from the left pane to the right pane.
 - ◆ **Selected User Roles:** The user roles in this pane will be assigned to the current user group. To remove a user role, drag it from the right pane to the left pane.
 - **Save:** Click to save a new user group and close the page.
 - **Clear:** Click to clear all the boxes.
 - **Save/Edit:** Click to save a new user group and display additional tabs.
 - **Cancel:** Click to cancel the operation.
 - **Copy Profile:** Opens a dialog box, where you can select a user group. All its settings will be copied. You can make the necessary changes.
3. Fill in the boxes of the page and click **Save**. The system saves the new user group to the database and displays the updated User Groups page, if the operation is completed successfully.

NOTE After you have created a user group, you can assign access rights to it. For details refer to [Assigning Access Rights](#).

Importing a User Group

You can import user groups from Active Directory of the user under which Monarch Server Agent Service is running. Use the **Import Groups** button for importing user groups.

The **Import Groups** button is available if Security Provider is selected from Active Directory and the Monarch Server Agent Service runs as user in Active Directory.

When a user from Active Directory logs in Monarch Server Admin or Client, the system starts checking all groups in the Active Directory in which the given user has membership. The checking is executed by the **sAMAccountName** parameter. The system checks whether the **sAMAccountName** parameter matches the **Alias** parameter of Monarch Server groups. In case of matches, privileges of all Monarch Server groups are granted to the given user during the current session only.

Editing a User Group

To edit a user group

1. On the Navigation Tree, click **Rights and Privileges**, and select **User Groups**.
2. Click the name of the user group you want to edit or click anywhere on the row, and then click . The User Group page appears.
3. Make the necessary changes. For details about how to change its items, refer to [Adding a User Group](#).
4. To grant access rights, select an appropriate tab at the top of the page. The following tabs are available in the **Edit** mode:
 - **Group** tab allows you to modify the user group name and description and grant common and specific privileges to the selected user group.
 - **Views** tab allows you to specify what [views](#) of the document search results page or RMS views page will be available for the current user group. To do this, select or clear appropriate check boxes, and click **Save**.
 - **Settings** tab allows you to modify the [notification settings](#), particularly the number of days when an alert about license expiration will be shown.

NOTE The **Defined** check box is disabled by default. If the setting is not “Defined,” then the global setting under System Preferences applies.

- **Portlets** tab allows you to [assign portlet access rights](#) to the current user user group..
- **Templates** tab allows you to assign document [templates](#) to the current user group. To do this, select a document type from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.
- **Models** tab allows you to assign [models](#) to the current user group. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
- **Automator Processes** tab allows you to assign Automator processes to the current user group. To do this, select a search from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.
- **Visual Processes** tab allows you to assign visual processes to the current user group. To do this, select a search from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.

NOTE Processes created in older versions of Monarch Server (e.g., v13.2) are assigned to all users.

- **Automator Projects** tab allows you to assign projects to the current user group. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
 - **Workspaces** tab allows you to assign workspaces to the current user group. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
 - **Change Lists** tab allows you to assign change lists to the current user group. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
 - **Folders** tab allows you to assign folders to the current user group. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
 - **Named Paths** tab allows you to assign named paths to the current user group. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
5. Make the necessary changes and click **Save**. The system saves the modified user group to the database and displays the updated User Groups page, if the operation is completed successfully.

Deleting a User Group

To delete a user group

1. On the Navigation Tree, click **Rights and Privileges**, and select **User Groups**.
2. Click  at the end of the row with the user group you want to delete. A dialog box appears.
3. Click **Delete** to delete the user group from the list.

USERS

Before customers can log on to Monarch Server, an administrator must add them to the list of users on the Users page, and specify their [access rights](#) and [privileges](#).

All users working in the system are listed on the Users page. A user is kept in the Monarch Server database and has no tie with other security systems.

To display the Users page, on the Navigation Tree, click **Rights and Privileges**, and select **Users**.

Use this page to set rights and privileges for users, as well as to [add](#), [edit](#), or [remove](#) users from the list.

An administrator can assign rights to users individually.

Adding a User

To add a user

1. On the Navigation Tree, click **Rights and Privileges**, and select **Users**.
2. Click  on the **Users** page. The **User** page containing the following items appears:
 - **User Name:** Enter a unique name of a new user. This name will be used as a logon name.
 - **Full Name:** This name will be displayed in all user lists and therefore must be unique. It is recommended that **User Name** NOT be the same as **Full Name** for security reasons.
 - **Password:** Type a password.
 - **Confirm Password:** Type the same password once again.
 - **Network Alias:** Select an Active Directory user name.

NOTE This field is available only if the [security provider](#) is set other than **Default**.

- **E-mail Address:** Enter an e-mail address of the current user.
- **User Class: Persistent or Dynamic.** This value is assigned automatically and displayed for reference only.
- **Locked:** Select this check box, if you want to lock the current user and prevent their access to the system.
- **Privileges:** By selecting check boxes, an administrator can grant the following privileges:
 - ◆ **Administrator:** Allows the user to log on to MSAdmin.
 - ◆ **Portlets:** Allows the user to view Panopticon Workbooks in MSCClient.
 - ◆ **Automator Administrator:** Allows the user full administration.
 - ◆ **Automator User:** Allows the user to manage processes and jobs.
 - **Can Edit Scripts:** Allows the user to edit process scripts and script items for Visual Processes and Standard Processes, as well as project scripts (i.e., PreProcess, PostProcess, and Global). If unselected, the user cannot edit the properties of script items for existing processes; the user can only run the script and check the results.
 - ◆ **Automator Light User:** Allows the user to partially manage processes and jobs.
 - **Can Edit Schedules:** Allows the user to add, edit, and delete schedules for Visual Processes and Standard Processes. Note that only the schedules whose processes are assigned to the user can be modified.
 - ◆ **RMS User:** Allows the user to connect to RMS Client.
 - ◆ **RMS Administrator:** Allows the user to log on to RMSAdmin.
- **Members of:** Click this heading to expand the hidden sections.
 - ◆ **Available User Groups:** Displays all available user groups from which you can select ones to assign the current user to. To add a user group, drag it from the left pane to the right pane.
 - ◆ **Selected User Groups:** The current user will be assigned to the user groups in this pane. To remove a user group, drag it from the right pane to the left pane.
 - ◆ **Available Roles:** Displays all available roles from which you can select ones to assign the current user to. To add a role, drag it from the left pane to the right pane.
 - ◆ **Selected Roles:** The current user will be assigned to the roles in this pane. To remove a role, drag it from the right pane to the left pane.

NOTE The default user groups and roles available in the list of available user groups and roles for the new user are those specified in the **New User Defaults** settings in **System Preferences** under **System Settings**. See [Groups and Roles](#).

- **Save:** Click to save a new user and close the page.
- **Clear:** Click to clear all the boxes.
- **Save/Edit:** Click to save a new user and display additional tabs.
- **Cancel:** Click to cancel the operation.
- **Copy Profile:** Opens a dialog box, where you can select a user. All its settings will be copied. You can make the necessary changes.

3. Fill in the boxes of the page, and click **Save**.
The system saves the new user to the database and displays the updated Users page, if the operation is completed successfully.

NOTE After you have created a user, you can assign access rights to it. For details refer to [Assigning Access Rights](#).

Changing User Password

As an MSAdmin, you can change an Monarch Server user's password at any time.

To change a user's password

1. On the Navigation Tree, click **Rights and Privileges**, and select **Users**.
2. Click the name of the user you want to change a password for or clicking anywhere on its row, and then click .
3. Type a password in the **Password** and **Confirm Password** boxes, then click **Save**.

NOTE If you are using a non-default [security provider](#), the password will be changed in the target security system.

Editing a User

To edit a user

1. On the Navigation Tree, click **Rights and Privileges**, and select **Users**.
2. Click the name of the user you want to edit or click anywhere on the row, and then click . The User page appears.
3. Make the necessary changes. For details about how to change its items refer to [Adding a User](#).
4. To assign a temporary password to a user, type a password in the **Password** and **Confirm Password** boxes, and click **Save**.
5. To grant access rights, select an appropriate tab at the top of the page. The following tabs are available:
 - **User** tab allows you to modify the user name, description, and login information and grant common and specific privileges to the selected user.
 - **Views** tab allows you to specify what [views](#) of the document search results page or RMS views page will be available for the current user. To do this, select or clear appropriate check boxes, and click **Save**.
 - **Settings** tab allows you to modify the [notification settings](#), particularly the number of days when an alert about license expiration will be shown.

NOTE The **Defined** check box is disabled by default. If the setting is not "Defined," then the

global setting under System Preferences applies.

- **Portlets** tab allows you to [assign portlet access rights](#) to the current user.
- **Templates** tab allows you to assign document [templates](#) to the current user. To do this, select a document type from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.
- **Models** tab allows you to assign [models](#) to the current user. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
- **Automator Processes** tab allows you to assign automator processes to the current user. To do this, select a search from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.
- **Visual Processes** tab allows you to assign visual processes to the current user. To do this, select a search from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.

NOTE Processes created in older versions of Monarch Server (e.g., v13.2) are assigned to all users.

- **Automator Projects** tab allows you to assign projects to the current user. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
- **Workspaces** tab allows you to assign workspaces to the current user. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
- **Change Lists** tab allows you to assign change lists to the current user. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
- **Folders** tab allows you to assign folders to the current user. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
- **Named Paths** tab allows you to assign named paths to the current user. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.

NOTE You can copy rights of a role, by clicking the **Copy Profile** button, available on the above mentioned tabs (except **Views** and **Report Security** tabs).

6. Make the necessary changes, and click **Save**.
The system saves the modified user to the database and displays the updated Users page, if the operation is completed successfully.

Deleting a User

To delete a user

1. On the Navigation Tree, click **Rights and Privileges**, and select **Users**.
2. Click **X** at the end of the row with the user you want to delete. A dialog box appears.
3. Click **Delete** to delete the user from the list.

SYSTEM SETTINGS

In this section you will learn how to change:

- ❑ [System preferences](#)
- ❑ [Application settings](#)
- ❑ [Information to be displayed in the "About" dialog box](#)
- ❑ [System default settings](#)
- ❑ [Default settings for a new user](#)
- ❑ [Global user settings](#)
- ❑ [PDF default print settings](#)
- ❑ [Automator settings](#)
- ❑ [Server locations settings](#)
- ❑ [RMS Settings](#)
- ❑ [Exception Dates](#)
- ❑ [Bulk Schedules Edits](#)

SYSTEM PREFERENCES

To access the **System Preferences** page, on the Navigation Tree, click **System Settings**, and select **System Preferences**.

The following tabs are available on the **System Preferences** page:

- ❑ **[System Preferences](#)**: Set system preferences.
- ❑ **[Application Settings](#)**: Manage e-mail, offline results, security and storage settings.
- ❑ **[About Information](#)**: Specify product and company information, which is displayed in the **About** dialog box.
- ❑ **[Time Intervals](#)**: Set the calendar week for running processes.
- ❑ **[New User Defaults](#)**: Set default parameters that will be applied to a new user.
- ❑ **[Global User Settings](#)**: Set global parameters that will be applied to every user.
- ❑ **[PDF Default Print Settings](#)**: Set default PDF parameters that will be applied to every user.
- ❑ **[Automator Settings](#)**: Set Automator parameters.
- ❑ **[Locations Management](#)**: Set your server location preferences.
- ❑ **[Named Path](#)**: Manage named paths.
- ❑ **[RMS Settings](#)**: Set RMS settings.
- ❑ **[Exception Dates](#)**: Set Exception dates.

On the **System Preferences** tab you can specify the following settings:

System Preferences

- Maximum transaction report size:** Define the maximum number of rows to read from the MS Transaction database.
- Dynamic search on lists:** Select this check box to enable the auto-suggest feature in MSCClient, when the user types an index value on the **Simple Search** page, or a report name on the Reports List or search criteria in all object lists in MSAdmin application.
- Display "data" items in the Administrator navigation tree:** The check box allows an administrator to show/hide "data" items on the Navigation Tree. As the amount of data may total up to hundreds of document groups, types, index fields, filing tasks and so on, it can be useful to hide all these items for easier navigation.
- Expand Administrator Tree:** Select this check box, if you want the MSAdmin Navigation Tree to be expanded to second-level navigation options by default.
- Hide advanced content:** Select this check box to hide some advanced options in the Navigation Tree and object settings.

NOTE No matter whether the **Hide advanced content** check box is selected or deselected, you can hide non-mandatory items by clicking the **Hide advanced options** heading inside the given section. To display them, click the **Show advanced options** heading.

- Implementation Name:** This optional name will be displayed on the Login page and on the **Welcome** line of the user interface.

PDF Security

- Enable secure PDF output:** Select this check box to disable editing and printing of the exported PDF document by client. By default this check box is cleared.
- Owner password:** Enables editing and printing of the exported PDF document only after entering the password. If no password is specified only export is allowed for all clients.
- User password:** Enables export of the PDF document only after entering the password. If no password is specified export is allowed for all clients.

Session Expiration Alert

- Admin timer of session expiration (minutes):** Specify the time of session expiration alert appearance before the end of session for MSAdmin application.
- Client timer of session expiration (minutes):** Specify the time of session expiration alert appearance before the end of session for Client applications.

Notification Settings

- Show alert before license expiration (days):** Specify when to show an alert about license expiration. The default setting is 30 days, i.e., you will see an alert banner at the bottom part of the page 30 days before your license expires. An alert icon will be visible on the top right portion of the page; it allows you to hide/display the alert banner.

NOTE No alert will be displayed when the field is set to "0".

This notification setting is not applicable to perpetual licenses.

The alert banner is color coded as follows:

- Gray: The license expires in more than a month.

- Orange: The license expires in less than a month but more than a week.
- Red: The license expires in less than a week.

- Save:** Saves system settings.
- Clear:** Clears all the boxes.
- Cancel:** Cancels the operation.

Application Settings

Use the Application Settings page to set your application default preferences. To access this page, on the Navigation Tree, click **System Settings**, select **System Preferences**, and click the **Application Settings** tab at the top of the page.

You can specify the following settings:

E-mail Settings

- Connection Type:** Select either **HTTP** or **HTTPS** from the list.
- Admin Host and Port:** Specify the name of the Monarch Server Admin Host and Port.
- Client Host and Port:** Specify the name of the Monarch Server Client Host and Port.
- Administrator E-mail Address:** Specify the ES administrator's e-mail address.
- Notification Sender:** Specify the e-mail address to appear in the **From** field of notification messages.
- E-mail Server User:** Specify the user name that is used for connection to the e-mail server.
- E-mail Server Password:** Specify the password that is used for connection to the e-mail server.

NOTE If no credentials are required on customers email system, then **E-mail Server User** and **E-mail Server Password** fields can be disabled by selecting the **Not Required** check box.

- E-mail Server Host:** Specify the name of the e-mail server used.
- E-mail Server Port:** Specify the port of the e-mail server.
 - **Use SMTPS:** Select this check box if you want to use SMTPS.
- Whitelist of recipient domains (e.g., "domain1.com, domain2.com"):** Specify the approved email domains.
 - **Use whitelist of recipient domains:** Select this check box if you want to use the whitelist of recipient domains.
- Test Email:** Click the button to test email settings.
- Send notification to system administrators when module terminated unexpectedly:** Select this check box if you want that admins to receive notification when the module terminates unexpectedly.
- Send notification to system administrators before license expiration:** Selecting this check box will notify system administrators with defined email addresses that the instance license expires in N days (as determined by the **Show alert before license expiration** setting in the **System Preferences** tab). The notification will be sent once per day.

NOTE A notification email will only be sent if the SMTP and email settings are correct (i.e., the email test is OK).

This check box is disabled by default.

- Send notification to system administrators when monitoring was disabled by an error:** Select this check box to notify users with common Admin privileges whenever monitoring is disabled due to an error. The notification will be sent via the users' email addresses.

Security Settings

- Permission Control**
 - **Do not include Global option on Rights tab:** A new “All Users Group” will be created automatically. This new user group will include all existing users.

NOTE Rights may be lost if the “All Users Group” is deleted.

- Only allow Automator Users to assign Rights to other users in their User Groups
- Include Global option on Rights tabs, for assignment of permissions to all users
 - **Automatically assign Global rights on new objects saved in the Server Library:** Global rights are automatically assigned to new objects only in the Server Library. They will not apply for processes.
- **Allow manual path selection:** Select this check box to make Local and Named Paths available everywhere. If the check box is not selected, the Local is hidden, and only Named Paths are available everywhere.
- Login attempts limit on Administrator:** Specify the maximum number of attempts to log in using the wrong username or password on MSAdmin.
- Login attempts limit on RMS:** Specify the maximum number of attempts to log in using the wrong username or password on RMSClient.
- Save:** Saves application settings.
- Clear:** Clears all the boxes.
- Cancel:** Cancels the operation.

About Information

To access the **About Information** page, on the Navigation Tree, click **System Settings**, select **System Preferences**, and click the **About Information** tab.

The **About Information** tab allows you to specify product and company information, which will be displayed in the **About** dialog box.

The **About Information** page displays the following text fields to be filled:

- Company:** Specify the company name.
- Phone:** Specify the phone number.
- Fax:** Specify the fax number.
- Mail:** Specify the e-mail address.

- Our Site:** Specify the website.
- Technical Support:** Specify the e-mail or website of technical support.
- Support Form:** Specify the e-mail address or website, containing the support form to be filled out by the customer.
- Save:** Saves system settings.
- Clear:** Clears all the boxes.
- Cancel:** Cancels the operation.

System Defaults

Use the **System Defaults** page to set your system default preferences. To access this page, on the Navigation Tree, click **System Settings**, select **System Preferences**, and click the **System Defaults** tab at the top of the page.

System defaults allow to specify values that will be used for the following parameters by default:

System defaults

- Default Locale:** From this list select a default locale for the Web Portal application. This locale will be displayed on the Login page and for all newly created users.

RMS System Defaults

- Default View:** Select the name of the view to show initially.
- Delete Models:** Select to remove source model after user session ends.
- Delete Reports:** Select to remove source report after user session ends.
- Delete Templates:** Select to remove source template after user session ends.
- Server-side XML/XSL transformation:** Select to use server XML/XSL conversion.
- Enable formatting after XML/XSL transformation:** Select to enable formatting for templates after XML/XSL conversion.
- Default Server Library Folder:** type in the name of a [Server Library Folder](#) to make its models appear in the **Selected Models** pane on RMS **My Home page** by default.
- Available Allowed Views:** The text box displays all available allowed data views that can be assigned for a particular user on RMS. You can shift the required data views into **Assigned Allowed Views** by clicking on them or dragging them. Use the **Add All** link to add all available allowed data views to the **Assigned Allowed Views** text box.
- Assigned Allowed Views:** The text box displays assigned allowed data views available for a particular user on RMS client. You can delete separate items in the view list by clicking on them. Use the **Remove All** link to clear the **Assigned Allowed Views** text box.
- Save:** Saves system default settings.
- Clear:** Clears all the boxes.
- Cancel:** Cancels the operation.
- Reset:** Resets all settings to default values.

Time Intervals

In the **Time Intervals** page, you can modify the calendar week settings to be used for running processes. To access this page, on the Navigation Tree, click **System Settings**, select **System Preferences**, and click the **Time Intervals** tab.

The following option can be modified:

- First day of each week:** From the drop-down, select the day on which a calendar week starts. The options include all days of the week.
- Save:** Saves system settings.
- Clear:** Clears all the boxes.
- Cancel:** Cancels the operation.

New User Defaults

The **New User Defaults** settings page define values that will be available for a new user by default. To access this page, on the Navigation Tree, click **System Settings**, select **System Preferences**, and click the **New User Defaults** tab at the top of the page.

The following settings can be customized for a new user:

SETTING	DESCRIPTION
Excel Output	<p>Use XLSX Excel spreadsheet format. Select this check box to use the XLSX format for Excel output.</p> <p>NOTE: This format requires Office 2007 installed on the client computer</p> <p>Request customize dialog. Select this check box to customize properties before data export.</p>
Report View	<p>Count pages The maximum number of documents that can be displayed on the Report View page at a time.</p> <p>Show greenbar Select this check box to enable alternate row highlighting when viewing report.</p>
Dynamic View	<p>Count table rows The maximum number of rows that can be displayed on the Dynamic View page.</p>
Data/Summary View	<p>Count table rows Specify the maximum number of rows that can be displayed on the Data View page.</p> <p>Count summary rows Specify the maximum number of rows that can be displayed on the Summary View page.</p> <p>Apply the style from the model Select this check box to apply the style from the model to the View page.</p> <p>Apply data formatting from model</p>

SETTING	DESCRIPTION
	<p>Select this check box to apply data formatting from the model to the View page.</p>
<p>CSV Output</p>	<p>Column Separator</p> <p>Specify the delimiter for CSV export from RMSClient and MSCClient</p> <p>Apply delimiter from model</p> <p>Select this check box to apply the delimiter from the model.</p> <p>Comma</p> <p>Select this button to use a comma as the delimiter for the CSV export.</p> <p>Semicolon</p> <p>Select this button to use a semicolon as the delimiter for the CSV export.</p> <p>Tab</p> <p>Select this button to use a tab as the delimiter for the CSV export.</p> <p>Pipe</p> <p>Select this button to use a pipe as the delimiter for the CSV export.</p> <p>Other</p> <p>Select this button to specify a preferred delimiter.</p>
<p>Locale Settings</p>	<p>Date Format</p> <p>Specify a date format for your searches. The following options are available: <i>mm/dd/yyyy</i> or <i>dd/mm/yyyy</i>.</p> <p>Decimal Separator</p> <p>Select one of the proposed decimal separators: comma or period.</p> <p>Group Separator</p> <p>Select one of the proposed group separators: comma, space or period</p>
<p>RMS Home Page Settings</p>	<p>Count home lists items: Default count of items on one page in Selected Reports, Selected Models, Selected Templates, and Model to Template Mappings Lists on My Home.</p> <p>Display report list: Select to display report list at RMS Home page</p> <p>Display model list: Select to display model list at RMS Home page</p> <p>Display template list: Select to display template list at RMS Home page</p> <p>Display model to template mapping list: Select to display template mapping list at RMS Home page</p> <p>Display news list: Select to display news list at RMS Home page</p>

SETTING	DESCRIPTION
Groups and Roles	<p>Available User Groups: Lists available user groups to be assigned to new user.</p> <p>Available Roles: Lists available roles to be assigned to new user.</p> <p>Once selected, the user groups and roles assigned to a new user are listed in the Selected User Groups and Selected Roles panes, respectively.</p>

- Save:** Saves new user default settings.
- Clear:** Clears all the boxes.
- Cancel:** Cancels the operation.

Global User Settings

Use the Global User Settings page to set your global settings. To access this page, on the Navigation Tree, click **System Settings**, select **System Preferences**, and click the **Global User Settings** tab at the top of the page.

The **Global User Settings** page contains the following settings:

- Maximum allowed report size (Kb):** Enter the maximum report size in kilobytes.
- Count of Items Per Page:** Define the number of items per page. The lists of Monarch Server Administrator are split into several pages. Select a value from the list. The default value is set to 10.
- Maximum exported CSV file size (rows):** Enter the maximum exported CSV file size in kilobytes.
- Maximum exported PDF file size (rows):** Enter the maximum exported PDF file size in kilobytes.
- Maximum exported PDF pages:** Enter the maximum number of exported PDF pages.
- Data View asynchronous loading limit (rows):** Number of rows set as a loading limit to be displayed as a result of exporting. The minimum value is 1000, the maximum value is 1000000 (one million), the default value is set to 5000.

If during report exporting number of Data View rows is less that the limit, all data will be exported immediately. If number of Data View rows exceeds the limit, export will be stopped on reaching the limit and the already exported data will be available for viewing. The rest of data will be exported in an asynchronous way.

The given feature allows you to speed up data exporting that are loaded within the defined limit.

After you made your changes

- To save changes to global user settings, click the **Save** button.
- To reset the changes you just made, click the **Clear** button.
- To cancel the operation without changes, click the **Cancel** button.

PDF Default Print Settings

Use the PDF Default Print Settings page to set your default settings for printing PDF files. To access this page, on the Navigation Tree, click **System Settings**, select **System Preferences**, and click the **PDF Default Print Settings** tab at the top of the page.

The **PDF Default Print Settings** page contains:

- PDF - Table Font Name:** Select the font name from the list.
- PDF - Report Font Name:** Select the font name from the list.
- PDF - Font Size:** Select any PDF font size from the list.
- PDF - Break Page Every:** Enter the number of rows. Zero value — no page break.
- PDF - Break Page Every:** Enter the number of pages. Zero value — no page break.
- PDF - Lines Spacing:** Select the value from the list.
- PDF - Readahead:** Specify the number of text lines to sample in order to determine paper type and orientation of PDF.
- PDF - Portrait Limit:** Specify the maximum number of characters per line for portrait paper orientation.
- PDF - Landscape Limit:** Specify the maximum number of characters per line for landscape paper orientation.
- PDF - Left Margin:** Specify the left margin size for PDF printing (in inches).
- PDF - Right Margin:** Specify the right margin size for PDF printing (in inches).
- PDF - Top Margin:** Specify the top margin size for PDF printing (in inches).
- PDF - Bottom Margin:** Specify the bottom margin size for PDF printing (in inches).

- Save:** Saves the settings.
- Clear:** Clears all the boxes.
- Cancel:** Cancels the settings.

Automator Settings

Use the **Automator Settings** page to set your Automator default preferences. To access this page, on the Navigation Tree, click **System Settings**, click **System Preferences**, and then select **Automator** tab.

The following tabs are available on the **Automator** page:

- [General Settings](#)
- [Custodian Schedules](#)
- [Automator Contacts](#)
- [Automator LDAP Settings](#)
- [Default Job Log Disposition](#)
- [Server Integration](#)
- [Input](#)

General Settings

Use the **General Settings** page to specify general **Automator** settings.

To access the **General Settings** page

1. On the Navigation Tree, click **System Settings**, and then select **System Preferences**.
2. At the top of the page, click **Automator**, and then select the **General Settings** tab.

To specify the **Prerequisite Command Line Timeout** settings

1. In the **Prerequisite Command Line Timeout** field specify a timeout for prerequisite command lines (in seconds). This tells Automator how long to wait before ending the command line and considering the prerequisite as failed. The default value is 10 seconds.
2. To save the settings, click the **Save** button.

To set the duration of workspace validation in Visual Processes

1. In **The number of seconds after which the workspace validation will be automatically canceled** field, set your desired duration of workspace validation when opening a Visual Process containing a workspace element.

NOTE Setting the value to "0" will disable workspace validation.

2. To save the settings, click the **Save** button.

To specify the Allow terminate Automator Jobs settings

Choose this option if you want to cancel the process export in progress.

1. Enable the **Allow terminate Automator Jobs** check box. The export is cancelled by clicking the **Cancel job** icon.
2. Disable the **Allow terminate Automator Jobs** check box. The export runs complete regardless of the job is cancelled.
3. To save the settings, click the **Save** button.

NOTE The **Allow terminate Automator Jobs** option is available for both standard and visual processes.

To specify the Allow ignore Global Alerts settings

Choose this option to allow global alerts to be ignored.

1. Enable the **Allow ignore Global Alerts** check box. The **Ignore Global Alert** check box becomes available in in the Process Alerts window.
2. Disable the **Allow ignore Global Alerts** check box. The **Ignore Global Alert** check box will not be available in the **Process Alerts** window.
3. To save the settings, click the **Save** button.

Custodian Schedule

The Custodian is responsible for implementing the [job log disposition settings](#), which tell Automator what to do with the job logs after the jobs have run.

After you specify job log disposition settings for a process, you need to specify a schedule for the Custodian, which is responsible for implementing the job log disposition settings. If you do not specify a schedule for the Custodian, the job log disposition settings will not be implemented.

To access the Custodian Schedules page

1. On the Navigation Tree, click **System Settings**, and then select **System Preferences**.
2. At the top of the page under **Automator Settings**, select the **Custodian Schedules** tab.

Use the Custodian Schedule page to [add](#), [edit](#) or [delete](#) the schedule for the job logs.

To suspend the running process, click the **Suspend** button.

To resume the suspended process, click the **Resume** button.

Adding a Custodian Schedule

After you specify job log disposition settings for a process, you need to specify a schedule for the Custodian, which is responsible for implementing the job log disposition settings. If you do not specify a schedule for the Custodian, the job log disposition settings will not be implemented.

To add a Custodian schedule

1. On the Custodian Schedules page, click the **Add** icon .
2. On the displayed page specify the following settings:
 - **Name:** Enter the name you want to assign to the schedule.
 - **Suspended:** Select this check box to suspend the task.
 - a. Select the task [execution frequency](#) from the following options: **One Time**, **Hourly**, **Daily**, **Weekly**, **Monthly**.
 - b. In the **At** box, set the time you want the process to run.
 - c. In the **On** box, set the start date of the process.
 - d. Select the appropriate timezone from the drop-down list.
 - e. Click the **Save** button, to save the schedule or **Run Now** to start the task immediately.

NOTE The Custodian schedule must be enabled for the job log disposition to occur at the specified time.

Editing a Custodian Schedule

When you open a saved Custodian schedule for editing, the following tabs become available:

- Custodian Schedule:** On this tab you can edit the schedule. For details about how to change the Custodian schedule, please refer to [Adding a Custodian Schedule](#).
- Settings:** On this tab you can specify the schedule settings.

The following settings are available:

- Delete the task if it is not scheduled to run again: Deletes the task after it has successfully run.
- Stop the task if it runs for n hour(s) n minutes: Specifies a run duration limit, which will stop the task after the specified number of hours and/or minutes has passed.

Deleting a Custodian Schedule

You can delete any schedule on the **Schedule** page.

NOTE The job log disposition settings will not be implemented, after you delete a schedule for the Custodian.

To delete a schedule

1. On the **Custodian Schedules** page click  at the end of the row with the schedule you want to delete. A dialog box appears.
2. Click **Delete** to remove the schedule from the list.

Automator Contacts

Use the Automator Contacts page to [add](#), [edit](#) and [delete](#) contacts.

To access the Automator Contacts page

1. On the Navigation Tree, click **System Settings**, and then select **System Preferences**.
2. At the top of the page under **Automator Settings**, select the **Automator Contacts** tab.

To add a new contact

1. On the **Automator Contacts** page, click the **Add** icon .
2. On the displayed page, specify the following information:
 - **Name:** Enter the contact name.
 - **Email:** Specify email.
3. Click the **Save** button.

To edit a contact

1. On the **Automator Contacts** page, do one of the following:
2. Click the name of the contact you want to change or click anywhere on its row, and then click .
3. Make necessary changes. For details see, [Adding a New Contact](#).

To delete a contact

1. On the Automator Contacts page, select the contact you want to delete.
2. Click  at the end of the row with the contact you want to delete. A dialog box appears.
3. Click **Delete** to remove the contact from the list.

Automator LDAP Settings

Use the **Automator LDAP Settings** page to configure LDAP Settings.

To access the Automator LDAP Settings page

1. On the Navigation Tree, click **System Settings**, select **System Preferences**, **Automator** tab, and then select **Automator LDAP Settings** tab.
2. Enter the following information
 - LDAP Path
 - User Name
 - Password
 - Filter Pattern
3. To save changes, click **Save**.

Default Job Log Disposition

Job log disposition settings specify what is to be done with the job log after a job is run, and under what circumstances the specified disposition will occur. Job log disposition settings can be specified on a [process basis](#) or you can specify default job log disposition settings, which will apply to all job logs.

NOTE After you specify job log disposition settings for a process, be sure to specify a schedule for the [Custodian](#) and enable it, as the Custodian is responsible for implementing the disposition settings. If you do not specify a schedule for the Custodian, the disposition settings will not be implemented.

To specify a default job log disposition

1. On the Navigation Tree, click **System Settings**, and then select **System Preferences**.
2. At the top of the page under **Automator Settings**, select the **Default Job Log Disposition** tab
3. Select one of the following Job Log Disposition actions:
 - **Leave:** Leaves the job log in its current location and with its current name.
 - **Remove:** Removes the job log.
 - **Move:** Moves the job log to the location you specify.
 - ◆ You can either enter the destination folder path in the text box or select the folder by clicking the **Browse** button. The **Named Paths** dialog box will appear.
 - ◆ From the Named Paths list, select the appropriate [named path](#).
 - ◆ Select the folder from the list of displayed in the Named Path Content box.
 - ◆ Click Save to add the folder.

NOTE You can also use a naming macro to assign a new name to the job log by selecting one from the drop-down list. To do this, click the **Insert Macro** icon  and select the desired macro.

4. Select one of the following Apply Disposition By Job Status settings:
 - **Completed:** Select this check box, if you want the specified disposition to occur when a job is successfully completed. Use the adjacent fields to specify the number of days and/or hours after the job is completed that you want the disposition to occur.
 - **Failed:** Select this check box, if you want the specified disposition to occur when a job fails to run successfully. Use the adjacent fields to specify the number of days and/or hours after the job fails that you want the disposition to occur.
 - **Cancelled:** Select this check box, if you want the specified disposition to occur when a job is cancelled. Use the adjacent fields to specify the number of days and/or hours after the job is cancelled that you want the disposition to occur.
5. Click the **Save** button to save current settings and close the dialog.

Server Integration

The DataPrep Export element in Automator's Visual Process Designer allows data to be exported in various formats. Before exporting Power BI file types, the API settings of the connector need to be modified. Use the Server Integration page to configure the settings for Power BI API.

To access the Server Integration page

1. On the Navigation Tree, click **System Settings**, and then select **System Preferences**.
2. At the top of the page under Automator Settings, select the **Server Integration** tab.

For the settings of Power BI, the following details should be provided:

- Client ID:** It is a string that carries user information as provided during registration in Power BI. It is used by these connectors to identify Automator once it is registered. The Client ID is public information that goes hand in hand with the Client Secret Key.
- Client Secret Key:** It is used to authenticate Automator when it requests access to user accounts in Power BI. It goes hand in hand with the Client ID and should be kept private.

NOTE To get the Client ID and Client Secret Key for **Power BI**, click [here](#).

- Host Name:** It is the full name of the machine on which Automator is installed.
- Port:** It is the value of the port on which Automator is installed.
- Redirect URI:** It is the URI to which Power BI directs after authenticating the user. The field is automatically filled after supplying the Client ID, Client Secret Key, and host name.
- Other settings
 - **Save:** Click this button to save the application settings.

Clear: Click this button to remove the information in the fields.

Power BI API Keys

To get a Client ID and Client Secret Key for Power BI

1. Access Power BI's Registration Tool at <https://dev.powerbi.com/apps>.

Microsoft | Power BI

Developer | App registration tool

Register your application for Power BI

Register your application with Azure Active Directory (Azure AD). You'll be able to manage and modify these settings later in the Azure portal. [Learn more](#)

STEP 1
Sign in to Power BI

Sign in to your Power BI account to start registering your application.

[Sign in](#) Don't have an account? [Sign up now, for free](#)

STEP 2
Register your application

NOTE The registration process in Power BI requires an Azure Active directory, an organizational user, and a Power BI service account. To sign up for an Azure subscription or to log in using an account with such subscription, click [here](#).

To learn how set up your Azure Active Directory and organizational user, click [here](#).

2. Sign in using your Power BI account in Step 1 of the Registration Tool.
3. Provide the following details:
 - a. **App Name** – The name of your app, i.e., Automator.
 - b. **App Type** – Choose “Server-side Web app.”
 - c. **Redirect URL** – This provides Power BI with more details about the app requesting the authentication. An example of a redirect URL is `http://<hostname>/MSAdmin/ProcessDesignerEditor/PowerBIRedirect`.
 - d. **Home Page URL** – Use the sign in page of Automator, i.e., [http://<hostname>/MSAdmin](#).

STEP 2
Register your application

Register your application with Azure AD to allow your application to access the Power BI REST APIs and to set resource permissions for your application. You can change this later in the Microsoft Azure portal. [Learn more](#)

Application Name
 Enter a display name to identify your application in Azure

Automator

Application Type
 Choose the type of application you are developing

Server-side web application (for web apps or web APIs)

Home Page URL
 Enter your application's homepage URL

http://<hostname>/MSAdmin

Redirect URL
 Enter a URL where users will be redirected upon sign in so your application can receive an authorization code.

http://<hostname>/MSAdmin/ProcessDesignerEditor/PowerBIRedirect

4. Select the APIs you need as well as the level of access required. For details about permissions in Power BI, click [here](#).

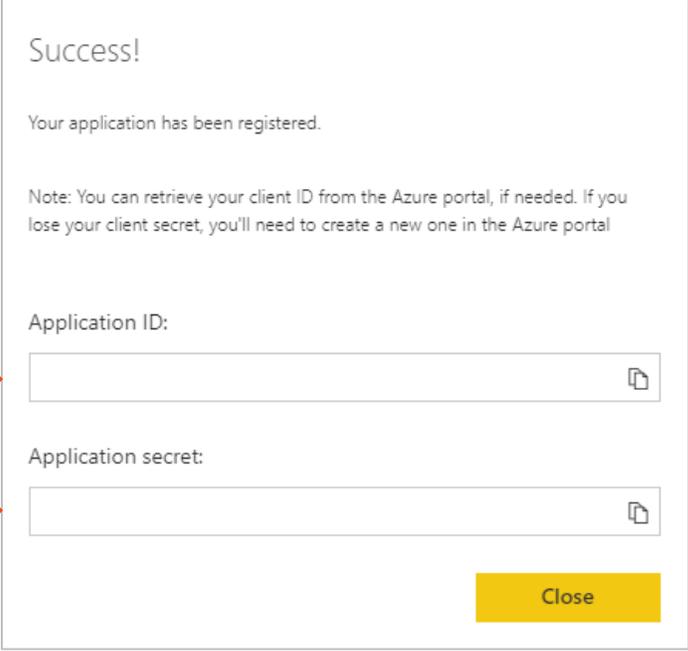
API access
 Select the APIs and the level of access your application needs. You can change these settings later in the Azure portal. [Learn more](#)

Select all

Read only APIs ⓘ	Read and write APIs ⓘ	Create APIs ⓘ
<input type="checkbox"/> Read all datasets	<input type="checkbox"/> Read and write all datasets	<input type="checkbox"/> Create APIs
<input type="checkbox"/> Read all dashboards	<input type="checkbox"/> Read and write all dashboards	
<input type="checkbox"/> Read all reports	<input type="checkbox"/> Read and write all reports	
<input type="checkbox"/> Read all workspaces	<input type="checkbox"/> Read and write all workspaces	
<input type="checkbox"/> Read all capacities	<input type="checkbox"/> Read and write all capacities	
<input type="checkbox"/> Read all storage accounts	<input type="checkbox"/> Read and write all storage accounts	
<input type="checkbox"/> Read all dataflows	<input type="checkbox"/> Read and write all dataflows	
<input type="checkbox"/> Read all gateways	<input type="checkbox"/> Read and write all gateways	
<input type="checkbox"/> Read all Power BI apps		

5. Click Register App.

The Registration Tool generates a Client ID and a Client Secret Key. Use these keys to configure the API settings for Power BI in Automator.



The screenshot shows a 'Success!' dialog box with the following content:

- Success!**
- Your application has been registered.
- Note: You can retrieve your client ID from the Azure portal, if needed. If you lose your client secret, you'll need to create a new one in the Azure portal
- Application ID: [Text input field with a copy icon]
- Application secret: [Text input field with a copy icon]
- Close [Yellow button]

Two red arrows point to the 'Application ID' and 'Application secret' input fields.

Input

Use the **Input** page to set the default Excel Import Engine to use in Automator.

To access the Input page

1. On the Navigation Tree, click **System Settings**, and then select **System Preferences**.
2. At the top of the page, click **Automator**, and then select the **Input** tab.

To set the default Excel Import Engine

1. In the Default engine to use for importing from Excel drop-down list, choose Legacy or Modern.
2. Select the Always substitute the Modern engine when executing a process which uses the Legacy engine checkbox to have all processes be executed using the Modern engine.
3. Click **Save** to save the settings.
4. Click **Clear** to cancel all changes made.
5. Click **Cancel** to cancel all changes made and return to the **General Settings** tab.

Locations Management

To manage your location preferences

1. On the Navigation Tree, click **System Settings** and select **System Preferences**.
2. On the displayed page select the **Locations Management** tab.

To set the default location name for all the object that have location as their parameter

1. Select location name from the **Default Location** list.
2. Click the **Save** button.

To add new location

1. Click the **Add New Location** button  and enter the new location name in the **Location** field.
2. Click the **Save** button. The new location will appear on the list.

To edit location

1. Click the name of the location you want to edit or click anywhere on its row, and then click .
2. Make your changes.
3. To save your changes, click the **Save** button.

To remove location

1. Click  at the end of the row with the location you want to delete.
2. Click **Delete** to confirm.

NOTE In case of single server installation refer to the Configurator Guide in the [Monarch Server Installation Guide](#) to change location settings.

Named Paths

Use the **Named Paths** page to [add](#), [edit](#) and [delete](#) named paths.

To access the Named Paths page

1. On the Navigation Tree, click **System Settings** and select **System Preferences**.
2. At the top of the page select the **Named Paths** tab.

To add a new named path

1. Click the **Add** icon .
2. Enter a path name in the **Name** field.
3. Select the path group in the **Path Group** field.

Path Group options:

- **Common:** The named path is available to all items.
- **CopyMove Action:** The named path is available to the copy\move distribution action of Standard Processes.
- **Database Inputs for Standard Processes:** The named path is available to the database input in Standard Process.
- **Database Inputs for Visual Processes:** The named path is available to the database input in Visual Process.
- **DataPrep Export Files:** The Named Path is available when defining the DataPrep Export item in a Visual Process.

- **DataPrep Input Files:** The Named Path is available to the Workspace input (to replace input path) as part of the 'Replace Paths' option when a Workspace is opened in the Server Library. This is to help convert the initial desktop derived path to one that is defined as part of the Monarch Server environment.
 - **Distribution for Standard Processes:** The named path is available to global distribution paths.
 - **Distribution for Visual Processes:** The named path is available to distribution paths in Visual Processes.
 - **Export Report Files:** The named path is available to Report Export for Standard/Visual Processes.
 - **Export Table and Summary Exception Files:** The named path is available to Table/Summary Export exception files for Standard/Visual Processes.
 - **Export Table and Summary Files:** The named path is available to Table/Summary Export file for Standard/Visual Processes.
 - **Export Table and Summary Folders:** The named path is available to Table/Summary Export folder for Standard/Visual Processes.
 - **Filing Task:** The named path is available to filing task path.
 - **Google Analytics Key Files:** The named path is available to the workspace with Google Analytics data source (to replace Google Analytics Key Files in data source).
 - **Monitoring:** The named path is available to monitoring path in Standard/Visual Processes.
 - **Move Disposition:** The named path is available to job log disposition.
 - **Project:** The named path is available to project files in Standard Processes.
 - **Project Data Link Files:** The named path is available to data link files in the project (Standard Processes).
 - **Project Inputs.** The named path is available to text input in the project (Standard Processes).
 - **Project Models:** The named path is available to models in the project (Standard Processes).
 - **Project Prerequisites.** The named path is available to prerequisites in the project (Standard Processes).
 - **Publish Tableau Export Passwords:** The named path is available to the Publish to Tableau option (to select password).
 - **Publish Tableau Export TabCmd:** The named path is available to the Publish to Tableau option (to select TabCmd).
 - **RSS Action:** The named path is available to the RSS distribution action for Standard Processes.
 - **Script References.** The named path is available to the script (to specify some references).
4. Select location in the Location field.
 5. Select **Create folder allowed** to show the **Create New Folder** field that will allow the user to create a folder in the file system.
 6. Select **Browse from root** to make the whole file system of a selected location is accessible to the user
or
Enter the path in the **Path** field. In this case only files and folder within the path selected are accessible to the user.
 7. Click the **Save** button.

To edit a named path

1. Click the name of the named path you want to change or click anywhere on the row, and then click .
2. Make necessary changes. For details, see [To add a new named path](#).

To assign rights to a named path

1. Click the **Rights** tab found beside the **Named Paths** tab.
2. If you want the selected named path to be available to all users, select the **Global** checkbox.

3. Under **User Groups**, choose the user groups to which the named path should be assigned by dragging and dropping them from the **Available User Groups** pane to the **Selected User Groups** pane.
4. Under **Roles**, choose the roles to which the named path should be assigned by dragging and dropping them from the **Available Roles** pane to the **Selected Roles** pane.
5. Under **Users**, choose the users to which the named path should be assigned by dragging and dropping them from the **Available Users** pane to the **Selected Users** pane.
6. Click the **Save** button to save all changes.

To delete a named path

1. On the **Named Paths** page select the project location you want to delete.
2. Click  at the end of the row with the named path you want to delete.
3. Click **Delete** to remove the named path from the list.

RMS Settings

Use this page to set RMS settings. To access this page, on the Navigation Tree, click **System Settings**, select **System Preferences**, and click the **RMS Settings** tab at the top of the page.

RMS Settings

SETTING	DESCRIPTION
RMS Settings	End session redirect url: The URL to redirect users in case of end of session.
Digital Signature Settings	<p>Certificate: Certificate for Excel projects.</p> <p>Certificate friendly name: Certificate friendly name for Excel exports.</p> <p>Enable digital signature: If it is selected and Certificate is not empty, Certificate will be available for Excel exports, else — Certificate will not be available even if Certificate value is defined.</p> <p>Force digital signature: Force all exports to XLSX with signature. This entry is used only if Certificate is defined.</p>
Navigation Settings	<p>Show About link: Select to show About Dialog in RMS Client.</p> <p>Show Preferences link: Select to show Preferences Dialog in RMS Client.</p> <p>Show Allow Xlsx Format: Select to show Excel format checkbox in RMS Client.</p> <p>Show Home: Select to show the My Home tab in RMS Client.</p> <p>Show My Models: Select to show the My Models tab in RMS Client.</p>

RMS Home Page Settings

SETTING	DESCRIPTION
---------	-------------

SETTING	DESCRIPTION
Home Page Text Font	The font for the customer text at the Home page.
Home Page Text Font Size	The font size for the customer text at the Home page.
Home Page Text Color	The color for the customer text at the Home page.
Home Page Graphic URL	Certificate friendly name for Excel exports.
Home Page Text	The customer text at the Home page.

RMS Security Settings

SETTING	DESCRIPTION
Hash algorithm	Hash algorithm that is used for checking checksum in the symmetric mode. One of the following: SHA1, SHA256, SHA284, SHA512, MD5.
Signing digest algorithm	Hash algorithm that is used for getting digest for verifying signature. One of the following: SHA1, SHA256, SHA284, SHA512, MD5.
Key algorithm	Signature algorithm. Private and public key must be generated with this algorithm. One of the following: DSA, RSA.
Encryption method type	Checksum validation method. One of the following: Assymetryc, Symmetric.
Expiration time (seconds)	Expiration time of checksum in seconds.
Ignore checksum	If selected, checksum generation and passing as a parameter in the initial request is ignored. Otherwise, it is required. Disabling checksum may be useful for testing, demonstration and development purposes — before production stage.

HECM Settings

SETTING	DESCRIPTION
File Downloader Url	The URL to the MECM FileDownloader servlet.
File Downloader Block Size (kb)	The maximum number of bytes to read in this retrieval. For example, a block size of 1048576 means: read the first 1 MB of the file. In the event that the file is larger than the block size requested by the client, the servlet will return an HTTP Partial Content response code (206). The

SETTING	DESCRIPTION
	client should follow up with another request to retrieve the next chunk of the file.
File Downloader Connection Timeout	The timeout while attempting to connect to the MECM FileDownloader servlet after which the connection is terminated, and an error is produced.

- Save:** Saves the settings.
- Clear:** Clears all the boxes.
- Cancel:** Cancels the settings.

Exception Dates

Use the **Exception Dates** page to [add](#), [edit](#) and [delete](#) exception rules for disabling the running of certain processes at specific dates (e.g. holidays like New Year celebrations).

NOTE At the date with a set exception, user will get a notification with a list of processes that will not be run that day.

To access the Exception Dates page

1. On the Navigation Tree, click **System Settings** and select **System Preferences**.
2. At the top of the page select the **Exception Dates** tab.

To add a new exception date

1. Click the **Add** icon .
2. Enter an Exception Date name in the **Name** field.
3. Enter a Description.
4. Select the **Suspended** checkbox if you want to suspend the current Exception Date.
5. Choose **Affected Processes** by selecting the respective checkboxes.
6. For each affected process type (e.g., Standard Processes and Visual Processes), you may assign any or all of the available processes to the exception date.
 - a. Click on the process type to open the list of available processes.
 - b. Drag the preferred process to be assigned from the left panel to the right panel or click the add button .
 - c. To remove an assigned process, drag it from the right panel to the left panel or click the delete button .

NOTE You may also select all processes available for all process types by clicking **Select All** or remove all selected processes for all process types by clicking **Select None**.

If all available processes are assigned by clicking the **Add** button or **Add All** button,

then all new processes will be added to the Available Processes list and will not be assigned to the exception date.

If all available processes are assigned by clicking the checkbox or **Select All** button, then all new processes will be assigned to the exception date.

7. Select the **Postpone Process execution** checkbox if you want to execute the process immediately after the exception date. To let the process run on the set schedule, leave this option unchecked.
8. Set up **Exception Frequency** by either selecting a single date or indicating a daily, weekly, or monthly occurrence.

In setting up the Exception Frequency, you can set all-day occurrence or set a specific timeframe.

NOTE When setting a monthly occurrence, you can select the Business Day option. A business day is considered every official work day of the week; another common term is work(ing) day. These are the days between and including Monday through Friday.

9. Click the **Save** button.

To edit an exception date

1. Click the name of the exception date you want to change or click anywhere on the row, and then click .
2. Make necessary changes. For details, see [To add a new exception date](#).

To delete an exception date

1. Click  at the end of the row with the exception date you want to delete.
2. Click **Delete** to remove the exception date from the list.

LOCALE MANAGEMENT

A locale is a group of settings that determine the language, time format, date format, currency symbol, and so on. The existing locales are listed on the Locale Management page.

To access the Locale Management page, click **System Settings** on the Navigation Tree, and select **Locale Management**.

Use this page to edit locales.

The Locale Management page contains the following items:

- Locale group:** Select from the list of options.
 - Select **Admin** if you need to edit labels appearing only in MSAdmin.
 - Select **Client** if you need to edit labels appearing only in MSClient.
 - Select **Common** if you need to edit labels appearing both in MSAdmin and MSClient.
 - Select **Service** if you need to edit labels appearing in System Reports.
 - Select **Database** if you need to edit labels appearing in the database.
- Locale Language:** Select a language from the list. This list includes languages that were selected during the installation.

- ❑ **Search:** You can search for a label, by typing its name in this box.
- ❑ **Constant name:** The list displays names of fields as they appear in code, to facilitate tracking their location.
- ❑ **Original name:** The list displays all labels appearing on the user interface.
- ❑ **New Name:** The list displays all labels appearing on the user interface after localization.
- ❑ **Reload** icon : Click this icon to reload the grid.
- ❑ **Edit** icon : To edit a label, click anywhere on the row and click this icon. The **Edit Record** dialog box appears. Type the new name into a box and click **Submit**.

NOTE You can open the **Edit Record** dialog box by clicking anywhere on a row.

You can edit labels in one of the following ways:

- One by one:
 - ◆ Click the name of a label.
 - ◆ Type a new name in the **Edit Record** dialog box.
- All at once:
 - ◆ Select the **Import Resources** tab.
 - ◆ After editing, click the **Upload** button to browse for an .xml file containing the list of all localized labels. Labels from the selected file will be uploaded and displayed on the Locale Management page.

NOTE When necessary, company logos can be changed in Locale Management. Different logos can also be added for different locations (English, Deutsch, Français).

To do so, modify the following constants under the **Common Locale Group**.

- **Theme_LoginPageLogoUri:** To change the logo in the login page
- **Theme_HeaderLogoUri:** To change the header logo
- **Theme_ShortcutIconUri:** To change the shortcut icon

To download the edited labels, click the **Export Resources** tab.

The Export Resource page contains the following items:

- **Locale Language:** Select a language from the list. This list includes languages that were selected during the installation.
- **Export Resources:** Click this button to open or download an .xml file, containing the list of all labels.
- **Export only overridden resources:** Select this check box, to export only overridden resources.

BULK SCHEDULES EDITS

The **Bulk Schedules Edits** feature allows you to apply bulk editing of schedules to Automator Standard Processes, Visual Processes, Pump Custodian Processes, and Maintenance Processes.

To use bulk schedule editing

1. On the Navigation Tree, click **System Settings**, and then select **Bulk Schedules Edits**.
2. On the displayed page, select the **Find a Schedule** tab.
3. In the **Select Process Type** field, select the process you want to edit from the drop-down list. The available options are:
 - Automator Standard Processes
 - Visual Processes
 - Pump Custodian Processes
 - Maintenance Processes
4. Select the **Include Suspended** check box, if you need the suspended schedules to be displayed.
5. In the **Search Criteria** section, specify the time parameters of the schedule you want to change.

You can get onscreen all schedules of the selected type by clicking the **All** button. You can also get onscreen the schedules with a certain value by selecting **Schedule** or **Next Execution Time**.

NOTE In the Search and Next Execution Time criteria, you can leave the fields empty to display all corresponding schedules.

Click **Find**.

The list of corresponding processes is displayed.

6. Select the **Replace a Schedule** tab.
7. In the **Execution Frequency** section, specify the time parameters of the schedule you want to change.
8. Specify the processes you want to re-schedule by enabling corresponding check boxes.
9. Click **Replace** to apply your changes.

If you need to suspend or resume any schedule, you can select it in the list and click the **Suspend/Resume selected Schedules** icon.

You can also create a new schedule for Automator Standard Process and Visual Process.

To create a new schedule

1. Select the **Create a Schedule** tab.
2. In the **Select Process Type** field, from the drop-down list select the process type for which you want to create a schedule.
3. Select the **Suspended** check box, if a new schedule must be suspended.
4. In the **Execution Frequency** section, specify the time parameters of the schedule you want to create.
5. Select the necessary schedules in the list, and then click the **Save** button to create a new schedule.

NEWS

MSAdmin provides the News Message feature as a way to easily share information with other Monarch Server users. You can post news messages on the [All News](#) and [Global News](#) pages. After posting a message, you specify which Monarch Server users you want the message to be visible to. The users you specify will be able to view the message on the News page within Client.

There are three types of news:

- All News:** These are all the news used in the system. They are available to user groups or users having the appropriate rights. For more information refer to [All News](#).
- Expired News:** These are news that have expired. For details, see [Expired News](#).
- Global News:** Include both all news and expired news. They are available to all users. For more information refer to [Global News](#).

ALL NEWS

All news messages can be viewed by Monarch Server users or user groups having the appropriate rights.

To access the **All News** page, click **News** on the Navigation Tree, and select **News Management**.

Use the All News page to [add](#), [edit](#) or [delete](#) a news message.

The All News page contains:

- Search:** Type a message content (or a part of it) in this box and press ENTER. The filter will be applied to the list of news messages, and only messages containing the specified characters will be displayed.
- News date:** Displays the dates when news messages were posted.
- Content:** Displays the news messages content.
- Expiration Date:** Displays the expiration dates of news messages. Expired dates are highlighted in red.
-  : Click to add a news message.
-  : Click to edit a message.

NOTE You can also open a message for editing by clicking its name on the list.

-  : Click to delete a message.
-  : Click to delete all expired messages from the database.

EXPIRED NEWS

The **Expired News** page lists news messages, for which the expiration date has arrived or has passed.

To access the **Expired News** page, click **News** on the Navigation Tree, select **News Management**, and click the **Expired News** tab at the top of the page.

Use this page to [add](#), [edit](#) or [delete](#) a news message.

The **Expired News** page contains:

- Search:** Type a message content (or a part of it) in this box and press ENTER. The filter will be applied to the list of news messages, and only messages containing the specified characters will be displayed.
- News date:** Displays the dates when news messages were posted.
- Content:** Displays the news messages content.
- Expiration Date:** Displays the expiration dates of news messages. Expired dates are highlighted in red.
-  : Click to add a news message.
-  : Click to edit a message.

NOTE You can also open a message for editing by clicking its name on the list.

-  : Click to delete a message.
-  : Click to delete all expired messages from the database.

GLOBAL NEWS

The **Global News** page lists all global news messages. Global news messages can be viewed by all Monarch Server users. To access the **Global News** page, click **News** on the Navigation Tree, select **News Management**, and click the **Global News** tab at the top of the page.

Use this page to [add](#), [edit](#) or [delete](#) a news message.

The **Global News** page contains:

- Search:** Type a message content (or a part of it) in this box and press ENTER. The filter will be applied to the list of news messages, and only messages containing the specified characters will be displayed.
- News date:** Displays the dates when news messages were posted.
- Content:** Displays the news messages content.
- Expiration Date:** Displays the expiration dates of news messages. Expired dates are highlighted in red.
-  : Click to add a news message.
-  : Click to edit a message.

NOTE You can also open a message for editing by clicking its name on the list.

-  : Click to delete a message.
-  : Click to delete all expired messages from the database.

ADDING A NEWS MESSAGE

To add a news message

1. On the Navigation Tree, click **News**, and select **News Management**. The **All News** tab is displayed by default.
2. Select the **Expired News** tab, if you want to add an expired message.
3. Select the **Global News** tab, if you want to add a global message.
4. Click  to open the News Management page. The News Management page contains the following items:
 - **News Date:** The current date is displayed by default. You can click the  icon and select another date.
 - **Expiration Date:** Click the  icon to specify the expiration date of the news.
 - **Is Global:** Select this check box, if you want to make the message available to all users.

NOTE If you leave this check box cleared, you will be able to grant access rights to certain users, user groups or roles. The **Rights** tab will become available after you save the new message and open it for editing.

- **News Text:** Type the text of your news message.
 - **Save:** Saves a new message.
 - **Clear:** Clears all the boxes.
 - **Cancel:** Cancels the operation.
5. Fill in the boxes of the page, and click **Save**. The system saves the new message to the database and displays the updated News Management page, if the operation is completed successfully.

EDITING A NEWS MESSAGE

To edit a news message

1. On the Navigation Tree, click **News**, and select **News Management**. The **All News** tab is displayed by default.
2. Select the **Expired News** tab, if you want to edit an expired message.
3. Select the **Global News** tab, if you want to edit a global message.
4. Click the name of the message you want to edit or click anywhere on its row, and then click .
5. Make the necessary changes. For details about how to change the items refer to [Adding a News Message](#).
6. To grant access rights, select the **Rights** tab at the top of the page.

NOTE This tab is available, if you leave the **Is Global** check box cleared while [creating a new message](#).

7. Expand the hidden sections, by clicking the **User Groups**, **Roles**, or **Users** heading. To select the roles, groups or users that you want to have rights to view the news, drag them from the left pane to the right pane.
8. Make the necessary changes and click **Save**. The system saves the modified message to the database.

DELETING A NEWS MESSAGE

To delete a news message

1. On the Navigation Tree, click **News**, and select **News Management**. The **All News** tab is displayed by default.
2. Select the **Expired News** tab, if you want to delete an expired message.
or
Select the **Global News** tab, if you want to delete a global message.
3. Click  at the end of the row with the message you want to delete. A dialog box appears.
4. Click **Delete** to delete the message from the list.

SYSTEM REPORTS

Monarch Server Administrator provides an easy means of generating system reports. The following types of system reports can be generated:

- ❑ [Configuration reports](#) provide information about the current configuration of various parts of the Monarch Server system, such as the existing roles, users, and so on.
- ❑ [Transaction reports](#) provide information about any actions that have occurred in the system, such as security, audit, and so on.

CONFIGURATION REPORTS

Configuration reports provide information about the current configuration of various parts of the Monarch Server system, such as the existing roles, users, and so on. You can generate the following types of configuration reports:

- ❑ [Product License](#): Displays the product licensee, the product expiration date (if any), etc.
- ❑ [Roles](#): Lists the existing roles and their assigned user groups and users.
- ❑ [User Groups](#): Lists the existing user groups, their assigned users, and their privileges.
- ❑ [Users](#): Lists the existing users, their e-mail addresses, and their locked privileges.
- ❑ [Security](#): Lists user's security attributes.
- ❑ [Automator Process Dependencies](#): Lists which projects are used by which Standard Processes, which models are used by which projects, and which models and workspaces are used by which Visual Processes.

To generate configuration reports

1. On the Navigation Tree, click **System Reports**, and select **Configuration**. The Configuration Reports page appears.
2. On the Configuration Reports page, click the link that corresponds to the desired report type.

Product License Report

A Product License report is one of the configuration reports provided by Monarch Server. This report includes the following licensing information:

- ❑ Name of the product
- ❑ Product vendor
- ❑ Product copyright information
- ❑ License expiration date
- ❑ Customer name
- ❑ Customer address
- ❑ Product Components (for example, Report View, Chart, and so on)
- ❑ Product options (for example, maximum allowed connections, machine names, and so on)

To view and print a Product License configuration report

1. On the Navigation Tree, click **System Reports** and select **Configuration**.
2. Click the **Product License** link. The Product License configuration report is displayed.
3. To print the report, click the  icon at the top of the page, then click **OK** in the **Print** dialog box.

NOTE For information on how to convert a report to another format, see [Converting a report to another format](#).

Roles Report

A Roles report is one of the configuration reports provided by Monarch Server. This report includes all existing roles, and provides the following information about them:

- Names of roles
- Privileges assigned to roles
- User groups assigned to roles
- Users assigned to roles

To view and print a Roles configuration report

1. On the Navigation Tree, click **System Reports** and select **Configuration**.
2. Click the **Roles** link. The Roles report is displayed.
3. To print the report, click the  icon at the top of the page, then click **OK** in the **Print** dialog box.

NOTE For information on how to convert a report to another format, see [Converting a report to another format](#).

User Groups Report

A User Groups report is one of the configuration reports provided by Monarch Server. This report includes all existing user groups, and provides the following information about them:

- Names of user groups
- Privileges assigned to user groups
- Users assigned to user groups

To view and print a user groups configuration report

1. On the Navigation Tree, click **System Reports** and select **Configuration**.
2. Click the **User Groups** link. The User Groups report is displayed.
3. To print the report, click the  icon at the top of the page, then click **OK** in the **Print** dialog box.

NOTE For information on how to convert a report to another format, see [Converting a report to another format](#).

Users Report

A Users report is one of the configuration reports provided by Monarch Server. This report includes all existing users, and provides the following information about them:

- User names assigned to users
- Full (or actual) names of users
- Privileges granted to users
- E-mail addresses of users

NOTE Information about full names and e-mail addresses is provided only if it was specified when an administrator added users to the system.

To view and print a Users configuration report

1. On the Navigation Tree, click **System Reports** and select **Configuration**.
2. Click the **Users** link. The Users report is displayed.
3. To print the report, click the  icon at the top of the page, then click **OK** in the **Print** dialog box.

NOTE For information on how to convert a report to another format, see [Converting a report to another format](#).

Security Configuration Report

A Security report is one of the configuration reports provided by Monarch Server. This report includes all existing users and their assigned rights, and provides the following information about them:

- User Name and internal ID
- Assigned user groups and roles
- Assigned security keywords

To view and print a Security configuration report

1. On the Navigation Tree, click **System Reports** and select **Configuration**.
2. Click the **Security** link. The Security configuration report is displayed.
3. To print the report, click the  icon at the top of the page, then click **OK** in the **Print** dialog box.

NOTE For information on how to convert a report to another format, see [Converting a report to another format](#).

Automator Process Dependencies

The Automator Process Dependencies Report is one of the Configuration Reports provided by Automator. This report presents which projects are used by which Standard Processes, which models are used by which projects, and which models and workspaces are used by which Visual Processes. It contains the following:

- Process ID
- Process name
- Host name
- Process type
- Models used
- Projects used
- Workspaces used
- Project name

To view and print an Automator Process Dependencies Report

1. On the Navigation Tree, click **System Reports** and select **Configuration**.
2. Click the **Automator Process Dependencies** link. The Automator Process Dependencies report is displayed.
3. To print the report, click the  icon at the top of the page, then click **OK** in the **Print** dialog box.

TRANSACTION REPORTS

Transaction reports provide information about any actions that have occurred in the system, such as security, audit, and so on.

You can generate the following types of transaction reports:

- [Security](#) — Lists the time, user name, and log on/log off information.
- [System](#) — Lists system status information (e.g., whether the e-mail system has been started, the sub-volumes have been mounted, etc.).
- [Added Users](#) — Lists information about users add during the date range selected.
- [Deleted/Suspended Users](#) — Lists information about users who have been deleted or suspended.
- [Users Not Logged In](#) — Lists information users who have not logged in over a date range.
- [Audit Entity Actions](#) — Lists information about audit entity actions.
- [Custodian Schedules Logs](#) — Lists information about custodian schedules events.
- [Dynamic Login Requests](#) — Lists information about dynamic login requests in RMS Client.

- ❑ [Automator Distribution Audit](#) – Lists all the export and distribution (move and copy) paths used in processes and all email distributions used in distribution actions for a specified period.

To view transaction reports

1. On the **Navigation Tree**, click **System Reports**, and select **Transaction**. The Transaction Reports page appears.
The Transaction Reports page contains the following items:
 - **Events From To:** Specify the beginning and the end of the time interval, for which you want to remove events.
 - **Clear Log:** Click this button to remove all events that occurred during the time interval defined earlier.
 - **Transaction Reports:** Contains links to transaction reports of different types.
2. To generate a report, on the Transaction Reports page, click the link that corresponds to the desired report type.

Security Report

A Security Report is one of transaction reports provided by Monarch Server. This report provides the following information:

- ❑ Date and time the user(s) log on to and log off from the system
- ❑ Name(s) of the user(s) who logged on to and log off from the system
- ❑ Date and time the user(s) log on to the system with incorrect password
- ❑ Name(s) of the user(s) who logged on to the system with incorrect password

To view and print a Security transaction report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **Security** link. The Security Transaction Report page is displayed.
3. In the **From ... To** fields specify a time period, for which you want to view the security actions. Click  to open the Calendar window, and select the date.

NOTE If you do not provide a date or time period, all actions for the specified user(s) will be displayed.

4. From the **User** list, select a user, whose actions you want to include in the report. (To view security information for all users, select **All Users**.)
5. Click **View**. The Security transaction report is displayed.
6. To print the report, click the  icon at the top of the page, then click **OK** in the **Print** dialog box.

NOTE For information on how to convert a report to another format, see [Converting a report to another format](#).

System Report

A System Report is one of transaction reports provided by Monarch Server. This report provides the following information:

- Date and time when each system action occurred.
- List of system actions that have occurred.

To view and print a system transaction report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **System** link. The System Report page is displayed.
3. In the **From ... To** fields specify a time period, for which you want to view system actions. Click  to open the Calendar window, and select the date.

NOTE If you do not provide a date or time period, all system actions will be displayed.

4. Click **View**. The System Transaction Report is displayed.
5. To print the report, click the  icon at the top of the page, then click **OK** in the **Print** dialog box.

NOTE For information on how to convert a report to another format, see [Converting a report to another format](#).

Added Users Report

Added Users Report is one of transaction reports provided by Monarch Server. This report provides the following information:

- Internal User ID.
- User name.
- Date when a user was added.

To view and print an Added Users transaction report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **Added Users** link. The **Added Users** page is displayed.
3. In the **From ... To** fields specify a time period, for which you want to view system actions. Click  to open the Calendar window, and select the date.

NOTE The maximum number of days is 90.

4. Click **View**. The Added Users Transaction Report is displayed.

5. To print the report, click the  icon at the top of the page, then click **OK** in the **Print** dialog box.

NOTE For information on how to convert a report to another format, see [Converting a report to another format](#).

Deleted/Suspended Users Report

A Deleted/Suspended Users Report is one of transaction reports provided by Monarch Server. This report provides the following information:

- User ID.
- User name.
- Date when a user was locked or suspended.

To view and print a Deleted/Suspended Users transaction report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **Deleted/Suspended Users** link. The Deleted/Suspended Users page is displayed.
3. In the **From ... To** fields specify a time period, for which you want to view system actions. Click  to open the Calendar window, and select the date.
4. Click **View**. The Deleted/Suspended Users Transaction Report is displayed.
5. To print the report, click the  icon at the top of the page, then click **OK** in the **Print** dialog box.

NOTE For information on how to convert a report to another format, see [Converting a report to another format](#).

Users Not Logged In Report

A Users Not Logged In Report is one of transaction reports provided by Monarch Server. This report provides the following information:

- Internal User ID.
- User name.
- Last logon date.

To view and print a Users Not Logged In transaction report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **Users Not Logged In** link. The Users Not Logged In page is displayed.
3. In the **From ... To** fields specify a time period, for which you want to view logon actions. Click  to open the Calendar window, and select the date.

4. Click **View**. The Users Not Logged In Transaction Report is displayed.
5. To print the report, click the  icon at the top of the page, then click **OK** in the **Print** dialog box.

NOTE For information on how to convert a report to another format, see [Converting a report to another format](#).

Dynamic Login Requests Report

A Dynamic Login Requests Report is one of transaction reports provided by Monarch Server Report Mining Edition. This report provides the following information on the RMS Client login:

- Report Paths.
- Model Paths.
- Template Paths.
- Templates to Models.
- Allowed Views.
- Default View.
- Admin Mode.
- Definition Group.
- Dynamic Filter.
- Locale.
- Export To.

To view and print a Dynamic Login Requests transaction report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **Dynamic Login Requests** link. The Dynamic Login Requests page is displayed.
3. In the **From ... To** fields specify a time period, for which you want to view logon actions. Click  to open the Calendar window, and select the date.
4. In the **User** drop-down list select the reported user.
5. Click **View**. The Dynamic Login Requests transaction report is displayed.
6. To print the report, click the  icon at the top of the page, then click **OK** in the **Print** dialog box.

NOTE For information on how to convert a report to another format, see [Converting a report to another format](#).

Audit Entity Actions Report

Audit Entity Actions Report is one of transaction reports provided by Monarch Server. This report provides the following information:

- Date and time when changes were made
- Name of the user(s) who made changes
- Category of the performed action (Create, Edit, Delete)
- Type of the changed entity

To view and print an Audit Entity Actions transaction report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **Audit Entity Actions** link. The Audit Entity Actions Report page is displayed.
3. In the **From ... To** fields specify a time period, for which you want to view the search actions. Click  to open the Calendar window, and select the date.

NOTE If you do not provide a date or time period, all search actions for the specified user(s) will be displayed.

4. From the **User** list, select a user, whose actions you want to include in the report. (To view search actions for all users, select **All Users**.)
5. From the **Event Category**, select the appropriate action. (To view all actions, select **All event categories**.)
6. From the **Entity Type**, select the type of the entity, changes to which you want to view. (To view all entities, select **All types**.)
7. Click **View**. The Audit Entity Actions transaction report is displayed.
8. To print the report, click the  icon at the top of the page, then click **OK** in the **Print** dialog box.

NOTE For information on how to convert a report to another format, see [Converting a report to another format](#).

Custodian Schedules Logs Report

Custodian Schedules Logs is one of transaction reports provided by Monarch Server. This report provides the following information:

- Internal Custodian Schedule ID.
- Custodian Schedule name.
- Custodian Schedule events.
- Date and time when a custodian schedule was started.

To view and print a Custodian Schedule transaction report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **Custodian Schedules Logs** link. The Custodian Schedules Logs page is displayed.
3. In the **From ... To** fields specify a time period, for which you want to view system actions. Click  to open the Calendar window, and select the date.

NOTE The maximum number of days is 90.

4. In the **Custodian Schedules** field, select the category from the list.
5. Click **View**. The Custodian Schedules Logs Transaction Report is displayed.
6. To print the report, click the  icon at the top of the page, then click **OK** in the **Print** dialog box.

NOTE For information on how to convert a report to another format, see [Converting a report to another format](#).

Automator Distribution Audit

Automator Distribution Audit is one of transaction reports provided by Monarch Server. This report presents all the export and distribution (move and copy) paths used in processes and all email destinations used in distribution for a specified period. The report provides the following information:

- Destination path/E-mail
- Process name (ID)
- Process type
- Host/Domain name

To view and print an Automator Distribution Audit report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **Automator Distribution Audit** link. The Automator Distribution Audit page is displayed.
3. In the **From ... To** fields, specify a time period for which you want to view system actions. Click  to open the Calendar window, and select the date.

NOTE If a date or time period is not provided, all destination paths and email destinations will be displayed.

4. To facilitate the search and filter out the results, enter desired keywords in any of the following:
 - Host or domain name
 - Destination path or E-mail
 - Process name

5. Click **View**. The Automator Distribution Audit Transaction Report is displayed.
6. To print the report, click the  icon at the top of the page, then click **OK** in the **Print** dialog box.

NOTE For information on how to convert a report to another format, see [Converting a report to another format](#).

CONVERTING A REPORT TO ANOTHER FORMAT

To convert a report to another format

- Select required format from the **Export to the selected format** list at the top of the page and click **Export**.
Formats available:
 - Acrobat (PDF) file
 - CSV (comma delimited)
 - Excel
 - Rich Text Format
 - TIFF file
 - Web Archive

ACTIVE USERS

Monarch Server provides the possibility to monitor active users in the system.

To access the Active Users Report, on the **Navigation Tree**, click **System Reports**, and select **Active Users**. The list displays the following information:

- User name
- Privileges
- Creation time
- Module name
- Terminate

Click the  icon in the **Terminate** column to close the current session for the selected active user. The **Terminate User Session** dialog box will pop up with the the following question: *Terminate selected User Session?*

Click **Delete** to confirm your choice or **Cancel** to remain the session.

TECHNICAL SUPPORT

Software Support Policy

For details about the latest product versions, product development, software releases, and maintenance and support policy, contact [Altair Support](#).

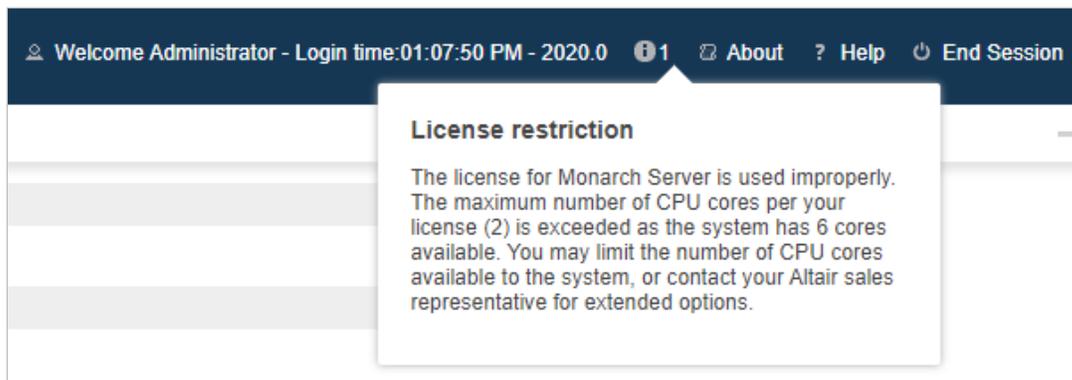
Before Contacting Technical Support

You have access to a variety of resources that can help you address any questions or concerns about Monarch Server. Before you contact Technical Support, do the following:

- ❑ Refer to the Monarch Server documentation. You may find the answer to your question in the help file.
- ❑ Check your configuration and resources. In some cases, a simple restart can solve the problem.
- ❑ Check your license, especially when you exceed the number of CPU cores allowed by your license, in which case the following message appears in the login page:

“The license for Monarch Server is used improperly. Please contact your Altair sales representative for extended options.”

A notification of your license restriction also appears on top of the main page of the Monarch Server Administrator.



You may refer to the [Server Status](#) page of **Automator** for further information.

How to Receive Help

- ❑ Access the [Altair Community](#).
- ❑ Email Altair Support at dasupport@altair.com.
- ❑ Call Altair Support (Monday–Friday).

US and Canada

Phone: +1-800-988-4739

Phone: +1-978-275-8350

Europe, Middle East, Africa

Phone: +44 (0) 8081 892481

NOTE Provide as much information as possible, including the following:

- Product name and version number
- Registration number
- Operating system and version number
- Exact error messages (where applicable)
- A description of the problem, including the steps necessary to reproduce it, and how you attempted to solve it
- Your contact information

What to Expect

We'll answer your call in the order we receive it. Here's what you can expect:

- If all support specialists are engaged with other customers, you will have the option to continue holding or to leave a message.
- The specifics of your call are logged into our call tracking system and queued for the appropriate technical support specialist.
- We will return your call as soon as a support specialist becomes available.
- Faxed requests are collected periodically throughout the day and then assigned to available support specialists.

GLOSSARY

Administrator: Any individual performing system administration, database administration, network administration, or warehouse administration duties.

Analytical Data: Data that has been evaluated, summarized or in some way analyzed.

Connection: A set of parameters necessary to connect to a data warehouse.

Copy: In Monarch Server Web, a storage location that will receive data files when documents are added to a storage group. A first copy (Copy 1) must reside on a mass storage device. Subsequent copies (Copies 2 and greater) may reside on any form of media.

Default Keyword: As opposed to a standard keyword, which is a value taken from a report, a default keyword is a single keyword that is automatically associated with every document created during a specific archive process or process job. It is used to mark or identify documents with a value that is not contained within the documents themselves. Use of default keywords is optional.

Document: A section of a report that is logically independent from other sections. Documents are the units that a report is divided into when it is stored in the report warehouse.

Document Break Field: An index field that is used to mark the beginning of a new document within a report. For example, a change in Purchase Order Number would be used to detect the start of a new document. The system administrator identifies the document break fields for a report for use during the archiving process.

Document Search: A database search for all documents that match the specified document type, document date range, and keywords. These criteria are specified in MSAdmin, while the document search itself is performed in Server.

Document Group: A document group is a classification of document types according to some common characteristic or subject. You could, for example, decide to group different kinds of financial reports into a single document group. Each document group contains one or more document types.

Document Template: A pre-designed document that contains formatting and, in many cases, generic text.

Document Type: A generic class of documents that have the same physical layout and function. Document types are assigned names and are classified into document groups by the system administrator.

Execution Frequency: Execution frequency (or run time) specifies both how often a process will run (e.g. once, hourly, daily, etc.) and at what time it will run. When scheduling a filing or maintenance process, you specify the execution frequency on the Filing Process page or the Maintenance Process page.

Filing: The process of indexing, paginating, and archiving incoming reports.

Filing Process: A process that executes all the filing tasks assigned to it.

Filing Task: A task that launches operations of indexing, paginating, and archiving incoming reports.

Global Security Keyword: A keyword associated with a role, user group or user that is logically added (the **AND** operation) to any retrieval requests. The purpose of a security keyword is to restrict user access to only those documents that contain this specific keyword.

Index Field: This is an internal pointer that associates a specific keyword with a specific document. It points to the start of a document, with which the given keyword is to be associated. Monarch Server uses indexes to locate and retrieve documents according to keywords. For example, when you type in the keyword **001-01-0001** for **Social Security Number**, the index is used to locate documents for the individual with that social security number.

Keyword: A specific value of a keyword type. **John Q. Public** is a keyword for the **Employee Name** keyword type.

Keyword type: A keyword type is the term, which is defined when you create a Monarch model (when creating a Monarch model, one or several keyword types are defined). This is a field within a model that identifies the document. For example, a keyword type for purchase orders might be **Purchase Order Number**. A keyword type for an employee report would be **Social Security Number**. A document type may have several keyword types associated with it. These keyword types are set up by a system administrator and are made available to users automatically.

Keyword value: This is the specific value of a keyword type. For example, **Q197-4112** could be a keyword for the **Purchase Order** keyword type.

Log Clearance Task: The process of clearing log-records from the Monarch Server database.

Locale: Defines one of the regional options. A locale affects the way some programs display and sort dates, times, currency and numbers.

Maintenance process: A process that executes all maintenance tasks assigned to it.

Maintenance Task: The process of copying, moving, and deleting files to maintain document storage.

Monarch Index Model File: A special form of Monarch Model File that contains only those fields that are used as keyword types for a given document type. A system administrator creates and controls this model file. It is used only during the archive process.

Monarch Model File: A file with the .dmod or .xmod extension, which contains field, filter, summary and chart definitions as well as other settings used by Monarch. A Monarch Model file is associated with each document type and is automatically opened when documents are viewed. A system administrator creates the default model file. End users can also create their own Monarch Model Files with their personal analysis settings and preferences.

Offline task: Offline processing provides the ability to process large datasets in the offline mode on a scheduled basis. If a user requests some operation (view report, table, summary, and so on) for a dataset, whose size exceeds the system limit, Monarch offers to process such requests in the offline mode. Schedule information for this task includes the start and stop time, which allows executing the task at the specified time only.

Online Mass Storage Device: Any storage device that provides immediate access to data.

Privileges: Permission to perform certain tasks within Monarch Server.

Process: An ordered sequence of tasks, scheduled by an administrator to perform once or periodically. It guarantees that tasks are performed sequentially, in proper order.

Removable Storage Device: Any storage device that requires human intervention to mount and unmount media when reading and writing data. A request for mounting a given storage group (or volume) must be issued before data can be retrieved or stored.

Report: A formal record containing information in a printed or electronic format. Reports have a consistent, structured layout and are generally stored in a single file on a storage volume.

Report Mining: The process of accessing and analyzing report data, usually across numerous reports contained in a report warehouse.

Report Premining: Pre-extracting table data from reports during a filing process (as part of archiving report data), and putting it into the MS database. The data will be used to build data views. No access to original report data is needed.

Report Warehouse: A repository containing reports that have been indexed for retrieval.

Rights: Permission to access any of the Monarch Server components, basic functions, as well as documents and custom queries.

Role: A role defines rights for users and user groups. It cannot include other roles. A role is kept in the Monarch Server database and has no tie with other security systems. An administrator can define rights via roles (Role Based Security) or assign them to user groups and individual users.

Search group: A set of logically related searches.

Search: Document retrieval criteria saved in the database under a particular name.

Security Keyword: A keyword associated with a role, user group or user that is logically added (the **AND** operation) to any retrieval requests. The purpose of the security keyword is to restrict user access to only those documents that contain this specific keyword.

Source: A set of parameters determining what data should be retrieved from a data warehouse.

Storage Group: The logical storage bins (referred to as storage groups) in which reports put into the Monarch Server system are stored. All the data in a storage group is stored together. Each storage group may be composed of several storage volumes.

Storage Volume: Volumes are used to segment the continually growing amount of data in a storage group. Each volume can have one or more copies, which may reside in separate storage locations. Several storage volumes comprise a storage group, each volume having a specific, user-defined size.

Task: Appropriate processing of a group of files from one incoming directory, described by task properties. Tasks have a name, description, and can be reused and assigned to multiple processes.

Text Search: A search through the text of documents for a given string of text. Documents must also meet the specified selections for document type, document date range, and index fields.

User: A user is an individual who has been granted product rights and privileges to Monarch Server by a system administrator.

User Group: A classification of users created to manage individual user's rights to the system.

User Name: A name or word defined by an administrator that lets Monarch Server recognize and identify authorized users. A user must supply a user name to log on to Monarch Server. Unique user names can be created for either each user, or for each type of user (for example, data entry clerk).

CONTACT US

GET IN TOUCH

We'd love to hear from you. Here's how you can [reach us](#).

SALES CONTACT INFORMATION

Portal: [Contact Altair](#)

US: + 1.800.445.3311

International: + 1.978.441.2200

SUPPORT CONTACT INFORMATION

Customer Portal: <https://community.altair.com/community>

Email: dasupport@altair.com

US: +1 800.988.4739

Canada: +1 978.275.8350

Europe, Middle East, Africa: +44 (0) 8081 892481