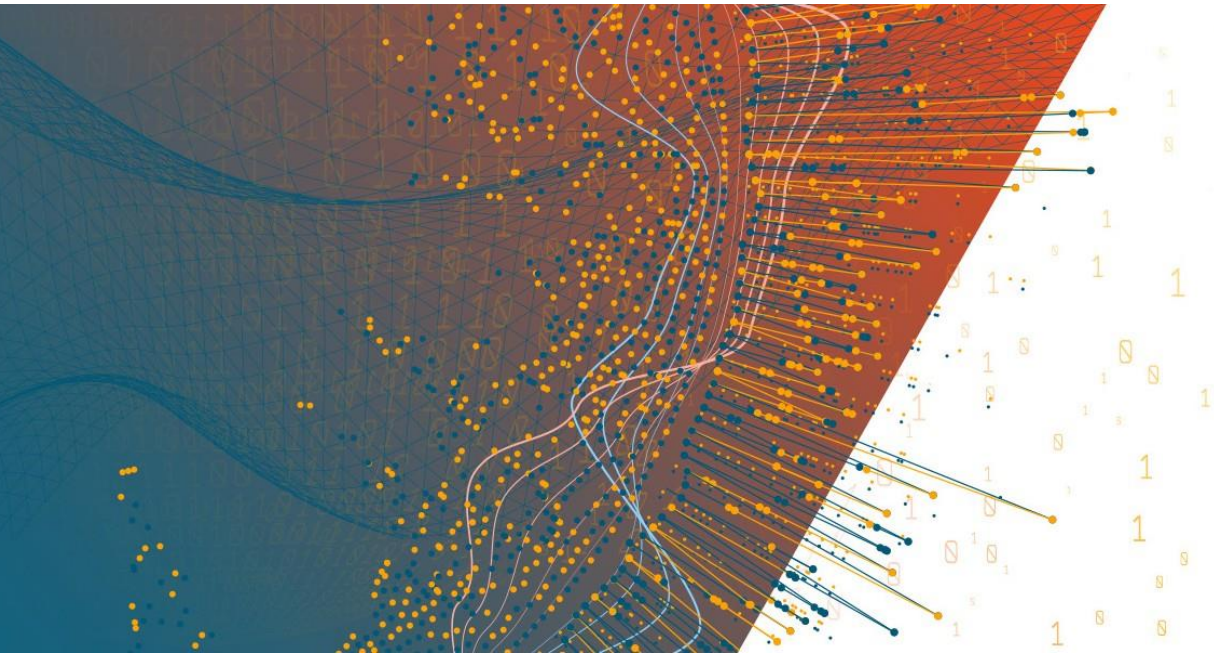


Altair® Monarch® v2021.1.1

MONARCH SERVER ADMINISTRATOR'S GUIDE

Report Warehouse | Automator

TABLE OF CONTENTS

Introduction **1**

About Monarch Server Automation Edition 1
Monarch Server Automation Edition Workflow 2
About Report Warehouse 3
Overview of MSAdmin Tasks 4
Launching Monarch Server Web Administrator 6
Navigation Tree 8
Functional Buttons 10
Page Navigation 11
Using the Calendar 12
Relational Operators 12
- Equal to (=) Operator 13
- Not Equal to (<>) Operator 13
- Less Than (<) Operator 14
- Greater Than (>) Operator 14
- Less Than or Equal to (<=) Operator 15
- Greater Than or Equal to (>=) Operator 15
- Boolean Operators 15

Report Warehouse **17**

Storage Management 17
- Storage Volumes 18
- Storage Groups 21
- Sub-Volumes 24

Maintenance Management 25
- Maintenance Tasks 25
- Log Clearance Tasks 35
- Maintenance Processes 37
- Full Text Search 41

Document Management 42
- About Models 42
- Index Fields 44
- Document Groups 49
- Document Types 51
- Filing Tasks 69
- Filing Processes 77

Document Retrieval 79
- Search Groups 80
- Searches 81
- Search Notifications 83
- System Notifications 85
- Filing Notifications 87

Offline Tasks 89
Offline Processes 89
Transfer 91
Export 91
Import 92

Automator 95

Server Status 95
Standard Processes 97
Processes 97
Schedules 115
Jobs 116
Distributions 118
Alerts 126
Global Environment Scripts 132
Project Editor 132
Visual Processes 151
Processes 152
Schedules 204
Process Templates 204
Process Logs 205
Alerts 206
Global Environment Scripts 209
Script Editor 209
PreExport Script 210
PostExport Script 210
PreProcess Script 211
PostProcess Script 211
Global Declarations Script for Standard Processes 211
Global Declarations Script for Visual Processes 212
References and Imports 212
Passwords 212
Inserting a Script for Standard Processes 212
Inserting a Script for Visual Processes 215
Testing a Script 216
Connection Definitions 217
Using the Naming Macros 218
Offset Expressions in Macros 222
When a Job Fails to Run 223
Runtime Fields 223
Project Inputs via Manifest 225
Project Outputs via Manifest 226
Standard Processes with Manifest Monitoring 229
Visual Processes with Manifest Monitoring 234
XSD for Standard Processes 239
XSD for Visual Processes 241

Portlets **252**

Portlets 252
Adding a Portlet 252
Editing a Portlet 253
Deleting a Portlet 254
Portlet Groups 254
Adding a Portlet Group 254
Editing a Portlet Group 255
Deleting a Portlet Group 255

Server Library **256**

Folders 256
Adding a Folder 256
Editing a Folder 258
Deleting a Folder 259
Projects 259
Adding a Project File 260
Editing a Project File 260
Opening a File in the Project Editor 262
Deleting a Project File 262
Changing a Project File State 263
Models 263
Adding a Model 263
Editing a Model 264
Deleting a Model 266
Changing Model State 267
Templates 267
Adding a Template 268
Editing a Template 268
Deleting a Template 270
Changing Template State 271
Workspaces 271
Adding a Workspace 271
Editing a Workspace 273
Deleting a Workspace 275
Changing Workspace State 275
Change Lists 276
Adding a Change List 276
Editing a Change List 277
Deleting a Change List 279
Changing Change List State 279
Bulk Edits 279

Rights and Privileges **281**

Assigning Privileges 281
SubAdmin 282
SaaS Administrator 282
Automator Administrator 283
Automator User 283
Automator Light User 283
Assigning Access Rights 284
Group Administrator 285
Setting Security Provider 287
About Security Keywords 288
Global Security Keywords 289
Assigning Security Keywords to Roles, User Groups or Users 290
Document States Security 291
Roles 291
Adding a Role 292
Editing a Role 293
Deleting a Role 295
User Groups 295
Adding a User Group 295
Importing a User Group 297
Editing a User Group 297
Deleting a User Group 299
Users 299
Adding a User 300
Changing User Password 301
Editing a User 302
Deleting a User 304
Document State Groups 304
Adding a Document State Group 304
Editing a Document State Group 304
Deleting a Document State Group 305
Document States 305

System Settings 306

System Preferences 306
Application Settings 308
About Information 310
System Defaults 311
Time Intervals 312
New User Defaults 312
Global User Settings 315
PDF Default Print Settings 317
Automator Settings 317
Locations Management 325
Named Paths 326
Exception Dates 328
Virtual Paths 329

Adding a Virtual Path 330
Editing a Virtual Path 330
Deleting a Virtual Path 330
Locale Management 330
Preprocess Converters 332
Bulk Schedules Edits 332

News 334

All News 334
Expired News 334
Global News 335
Adding a News Message 336
Editing a News Message 336
Deleting a News Message 337

System Reports 338

Configuration Reports 338
Product License Report 339
Storage Groups Report 339
Document Groups Report 340
Index Fields Report 340
Roles Report 341
User Groups Report 341
Users Report 342
Maintenance Tasks Report 342
Filing Tasks Report 343
Security Configuration Report 343
Model Indexes Configuration Report 344
Document Type Security Report 344
Automator Process Dependencies 345
Transaction Reports 345
Document Actions Report 346
Report Actions 347
Search Actions Report 348
Security Report 349
Filing Report 349
Maintenance Report 350
System Report 350
Reports Archived 351
Added Users Report 352
Deleted/Suspended Users Report 352
Users Not Logged In Report 353
Deleted Documents Report 353
Report Variance 354
Deleted Filing Results Report 355

Audit Entity Actions Report 355
Custodian Schedules Logs Report 356
Automator Distribution Audit 357
Annotation Reports 357
Annotations 359
Removed Reports 359
Converting a Report to Another Format 360
Activity Monitor 360
Active Users 360

Technical Support 362

Glossary 364

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 Warehouse

  Report Warehouse is a reliable platform for report and document content management. In Report Warehouse, reports and documents are "read-in" to a storage system and then indexed. These stored reports and documents can be easily retrieved using Report Warehouse's powerful query system and subsequently transformed into data views and Excel spreadsheets. The filing, storage, and retrieval of reports and documents are made easy in Report Warehouse.

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

- Monarch Server Automation Edition
- Automation Edition Workflow
- Monarch Server Report Warehouse
- MSAdmin Tasks
- Launching MSAdmin
- Navigation Tree
- Functional buttons
- Page Navigation
- Using the Calendar
- Relational Operators
- Boolean Operators

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.

ABOUT REPORT WAREHOUSE

Report Warehouse is a reliable platform for report and document content management. In Report Warehouse, reports and documents are "read-in" to a storage system and then indexed. These stored reports and documents can be easily retrieved using Report Warehouse's powerful query system and subsequently transformed into data views and Excel spreadsheets. The filing, storage, and retrieval of reports and documents are made easy in Report Warehouse.

At the heart of Monarch Server is Report Warehouse's report warehouse subsystem for efficiently storing and managing large volumes of corporate reports. Reports produced in any host environment can be brought into Report Warehouse. These reports are automatically identified, indexed, compressed, and stored in the warehouse system. Monarch Server uses an industry-standard SQL database to manage storage and provide fast, efficient access to report data.

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.

Report Warehouse Components

The MSAdmin is a part of the Report Warehouse system, which consists of these major components:

- **SQL database**. Report Warehouse works with industry-standard SQL databases. It supports Microsoft SQL Server 2016/2017/2018. 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.
- **Monarch Server Administrator (MSAdmin)**. It is used by the system administrator to define the parameters of the system. MSAdmin identifies where reports are to be found, how they are to be processed into electronic documents, and where the results (that is, documents) will be physically stored. It calls Monarch to set up models for indexing and extracting data from documents. MSAdmin provides capabilities for enforcing system security, establishing user rights and privileges, and handling maintenance issues.
- **Monarch Server Client (MSClient)**. It contains the functionality necessary for retrieving documents. It allows users to query the report warehouse in a number of ways to find the documents that they are looking for. Once found, these documents are launched into Monarch with the appropriate model file for viewing and analysis. MSClient is also used to process reports into documents. Using the parameters established in MSAdmin, new reports are recognized, indexed, broken into documents, and stored away, creating a warehouse of documents and indexes.

OVERVIEW OF MSADMIN TASKS

MSAdmin is the administration interface for Report Warehouse. Report Warehouse provides automatic indexing, instant retrieval of selected pages, and comprehensive security features.

In order to understand how Report Warehouse works, you need to understand its terminology. Although the terms are familiar, they have specific meanings in the context of Report Warehouse.

In Report Warehouse, reports are the electronic versions of the reports generated by your company's reporting systems. They have a consistent and structured layout, and they are generally stored as one file. For instance, an Employee Insurance Report is a report that might be stored in a single file called EMPINSUR.TXT.

A document is a section of a Report Warehouse report that can be logically independent from other sections of the report. The Employee Insurance Report might be sorted by office, with a subtotal for each department. Each section of the report that represents a regional department is logically independent of the others. Breaking the report into smaller logical documents allows users to retrieve only those pages of the report that they want.

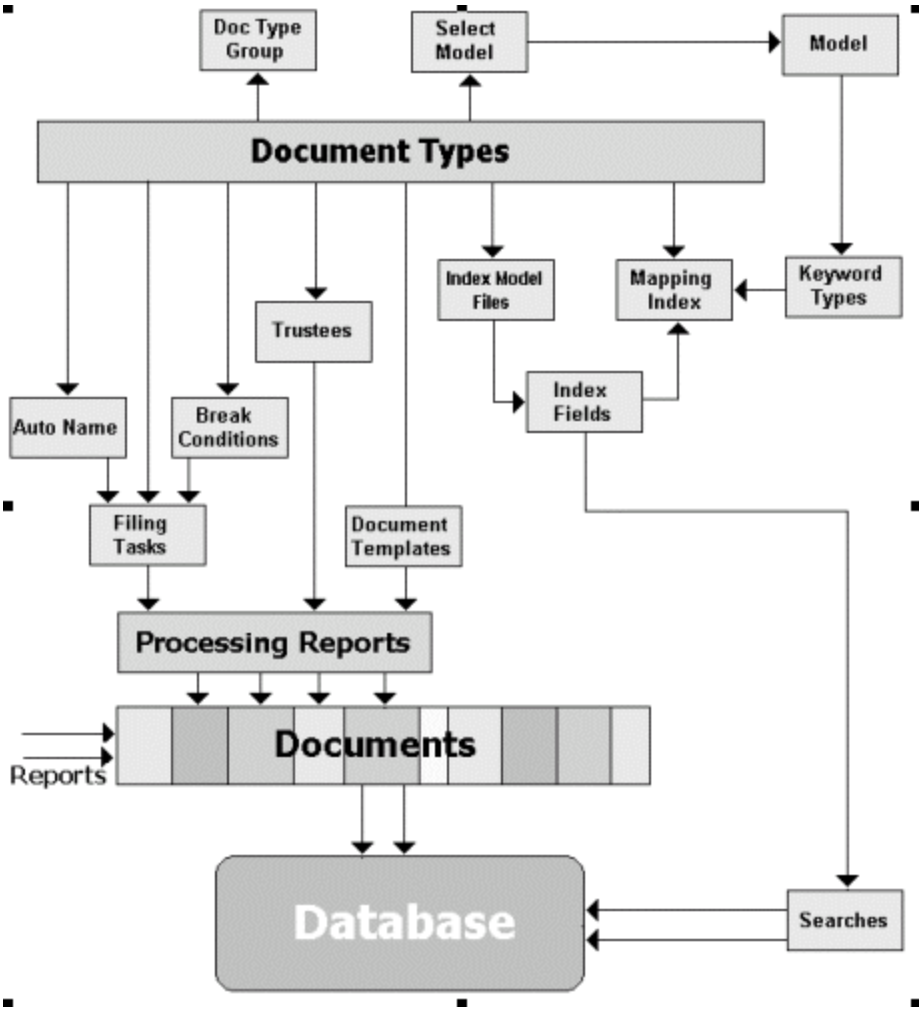
A keyword type is defined when you create a Monarch model, specified by a Monarch model file. In other words, when creating a model, one or several keyword types are defined. A keyword type is a field within a model that identifies the document. Document retrievals are based upon these fields of information.

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. While a keyword type is a general category, a keyword is the specific value of a keyword type. For example, 'Q197-4112' would be a keyword for the keyword type of Purchase Order Number. The keyword type of Social Security Number would contain the keywords '011-39-0778' and '287-33-1111'.

An index field is actually a keyword type but is kept in the database. When Monarch begins processing reports (filing), the system matches index fields with corresponding keyword types. All values of keywords for a document are saved in the database and are associated with index fields. A user can then begin searching for documents and setting appropriate values of index fields.

An index field points to the start of the document, with which the given keyword is to be associated. Report Warehouse uses index fields to locate and retrieve documents according to keywords. For example, when you type in the keyword '011-39-0778' for Social Security Number, the index is used to locate documents for the individual with that social security number.

The following scheme represents the main relationships between the parts of the system at the conceptual level.



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.

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.

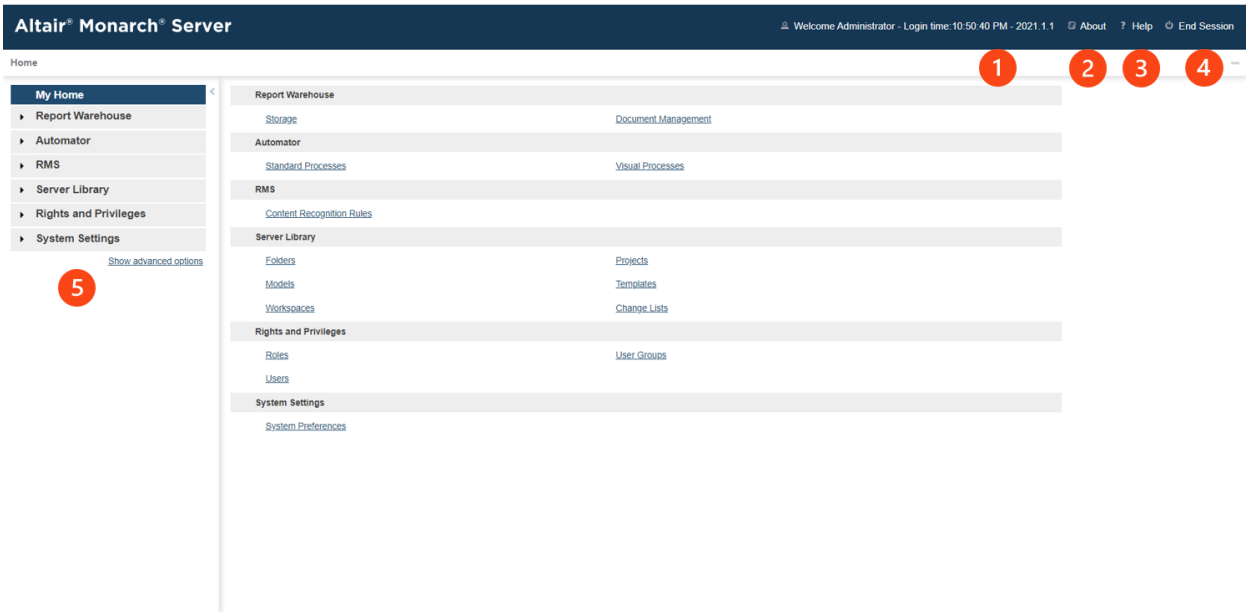
> **NOTE** In Monarch Server that is upgraded from previous versions (e.g., v13.2), the virtual folders could still be labeled as “DSAdmin” (or "DSClient"). In this case, use “DSAdmin” (or "DSClient”) instead of “MSAdmin” (or “MSClient”) to open the Admin (or Client) page.

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	**Login Details** This shows the last login time and the current version of the application.
2	**About** 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. **NOTE:** For Monarch Server 2021.1.1, the SDK and DataPrep versions are as follows: **SDK version**: 17.1.1.31052 **DataPrep version**: 17.1.1.31052
3	**Help** Clicking this button opens a related page in the Administrator's Guide.
4	**End Session** Clicking this button logs you out of the application.
5	**Navigation Tree** This panel serves as the navigation point for Monarch Server. Click **Show advanced options** to see all available sections in

Panels and Functions	Description
	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
  - Storage
    - **Storage Volumes**: This link allows you to manage storage volumes. For more information, see Storage Volumes.
    - **Storage Groups**: This link allows you to manage storage groups. For more information, see Storage Groups.
  - Maintenance
    - **Maintenance Tasks**: This link allows you to manage maintenance tasks. For more information, see Maintenance Tasks.
    - **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.
    - **Full Text Search**: This link allows you to search for reports containing the specified text.
  - Document Management
    - **Index Fields**: This link allows you to manage indexes. For more information, see Index Fields.
    - **Document Groups**: This link allows you to manage document groups. For more information, see Document Groups.
    - **Document Types**: This link allows you to manage document types. For more information, see Document Types.
    - **Filing Tasks**: This link allows you to manage filing tasks. For more information, see Filing Tasks.
    - **Filing Processes**: This link allows you to manage filing processes. For more information, see Filing Processes.
  - Document Retrieval
    - **Search Groups**: This link allows you to manage search groups. For more information, see Search Groups.
    - **Searches**: This link allows you to manage searches. For more information, see Searches.
    - **Search Notifications**: This link allows you to manage search notifications. For more information, see Search Notifications.
    - **System Notifications**: This link allows you to manage system notifications. For more information, see System Notifications.

        - **Filing Notifications**: This link allows you to manage filing notifications. For more information, see Filing Notifications.
        - **Offline Tasks**: This link allows you to manage offline tasks. For more information, see Offline Tasks.
        - **Offline Processes**: This link allows you to manage offline processes. For more information, see Offline Processes.
    - Transfer
        - **Export**: This link allows you to export document types and document-related items. For more information, see Export.
        - **Import**: This link allows you to import document types and document-related items. For more information, see Import.
- 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.
- 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.
- **Document State Groups**: This link allows you to manage document state groups. For more information, see Document State Groups.

❑ System Settings

- **System Preferences**: This link allows you to set system preferences. For more information, see System Preferences.
- **Virtual Paths**: This link allows you to manage virtual paths. For more information, see Virtual Paths.
- **Locale Management**: This link allows you to manage locales. For more information, see Managing Locales.
- **Preprocess Converters**: This link allows you to convert files of foreign, non-standard formats for processing them by the system. For more information, see Preprocess Converters.
- **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.
- **Annotations**: This link allows you to generate annotation reports. For more information, see Annotation Reports.
- **Removed Reports**: This link allows you to view the list of removed reports. For more information, see Removed Reports.
- **Activity Monitor**: This link allows you to monitor the filing and maintenance activities over a period of time. For more information, see Activity Monitor.
- **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

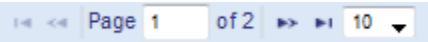
1. Click at the end of the row with the item you want to delete.
2. Click Delete to remove the corresponding item.

> **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:

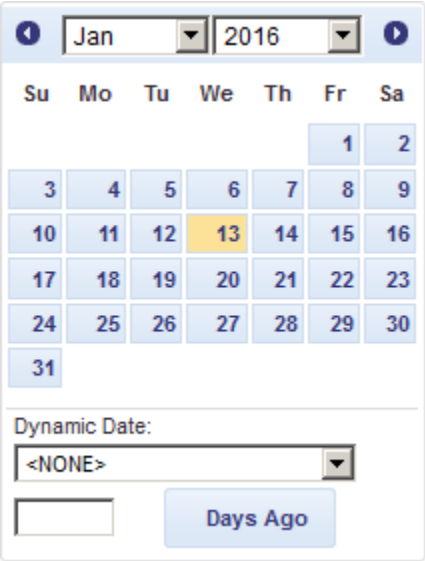
- To navigate to the next page, click .
- To navigate to the last page, click .
- To navigate to the previous page, click .
- To navigate to the first page, click .

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 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 .

RELATIONAL OPERATORS

Relational operators let you specify the relationship between an index field and an index value in your search criteria.

The relational operators are located in the **Operator** drop-down list, on the Search page.

> **NOTE** This list is not visible, until you select a document group and document type, and click the **Add condition** button under the **Search Criteria** heading.

The following relational operators are available:

- Equal to (=)
- Not equal to (<>)
- Less than (<)
- Greater than (>)
- Less than or equal to (<=)
- Greater than or equal to (>=)

When using relational operators in conjunction with Boolean operators, make sure that your search criteria make sense. For example, if your search criterion is **Employee Name = John Smith** and you add the search phrase **Employee Name = Jane Doe** to it, you cannot link the two phrases with the **AND** operator, because none of the **Employee Name** index fields would contain both **John Smith** and **Jane Doe** index values. (The **AND** operator specifies that *all* search phrases linked by it must be true.) To use these two search phrases in your search criteria, you would have to link them with the **OR** operator.

Equal to (=) Operator

When configuring a search, the equal to (=) operator is one of the relational operators you can use to specify the relationship between an index field and an index value in your search criteria.

Though the **equal to** (=) operator is the default relational operator used, you can select any of five other relational operators from the **Operator** drop-down list, on the Search page.

> **NOTE** This list is not visible, until you select a document group and document type, and click the **Add condition** button, under the **Search Criteria** heading.

Use the **equal to** (=) operator to retrieve only those documents that contain, in the specified index field, the index value contained in the search phrase.

Examples:

Customer Name = **Bob's Records** retrieves all documents with an index value of **Bob's Records** in the **Customer Name** index field.

Ship Date = **01/31/2001** retrieves all the documents with an index value of **01/31/2001** in the **Ship Date** index field.

Not Equal to (<>) Operator

When configuring a search, the **not equal to** (<>) operator is one of the relational operators that lets you specify the relationship between an index field and an index value in your search criteria.

The **not equal to** (<>) operator is located in the **Operator** drop-down list, on the Search page.

> **NOTE** This list is not visible, until you select a document group and document type, and click the **Add condition** button, under the **Search Criteria** heading.

Use the **not equal to** (<>) operator to retrieve only those documents that do not contain, in the specified index field, the same index value as the one in the search phrase.

Examples:

Customer Name <> **Betty's Music Store** retrieves all documents with index values other than **Betty's Music Store** in the **Customer Name** index field.

Ship Date <> **01/31/2001** retrieves all documents with index values other than **01/31/2001** in the **Ship Date** index field.

Less Than (<) Operator

When configuring a search, the **less than** (<) operator is one of the relational operators you can use to specify the relationship between an index field and an index value in your search criteria.

The **less than** (<) operator is located in the **Operator** drop-down list, on the Search page.

> **NOTE** This list is not visible, until you select a document group and document type, and click the **Add condition** button, under the **Search Criteria** heading.

Use the **less than** (<) operator in a search to retrieve only those documents that contain, in the specified index field, an index value that is less than the one in the search phrase.

Examples:

Ship Date < **06/01/2001** retrieves all documents with an index value of **05/31/2001** or earlier in the **Ship Date** index field.

Invoice Amount < **20000** retrieves all documents with an index value of **19,999.99** or less in the **Invoice Amount** index field.

Greater Than (>) Operator

When configuring a search, the **greater than** (>) operator is one of the relational operators you can use to specify the relationship between an index field and an index value in your search criteria.

The **greater than** (>) operator is located in the **Operator** drop-down list, on the Search page.

> **NOTE** This list is not visible, until you select a document group and document type, and click the **Add condition** button, under the **Search Criteria** heading.

Use the **greater than** (>) operator to retrieve only those documents that contain, in the specified index field, an index value that is greater than the one in the search phrase.

Examples:

Ship Date > **06/01/2001** retrieves all documents with an index value of **06/02/2001** or later in the **Ship Date** index field.

Invoice Amount > **20000** retrieves all documents with an index value of **20,000.01** or more in the **Invoice Amount** index field.

Less Than or Equal to (<=) Operator

When configuring a search, the **less than or equal to** (<=) operator is one of the relational operators you can use to specify the relationship between an index field and an index value in your search criteria.

The **less than or equal to** (<=) operator is located in the **Operator** drop-down list, on the Search page.

> **NOTE** This list is not visible, until you select a document group and document type, and click the **Add condition** button, under the **Search Criteria** heading.

Use the **less than or equal to** (<=) operator to retrieve only those documents that contain, in the specified index field, an index value that is less than or equal to the one in the search phrase.

Examples:

Ship Date <= **07/01/2001** retrieves all documents with an index value of **07/01/2001** or earlier in the **Ship Date** index field.

Invoice Amount <= **15000** retrieves all documents with an index value of **15,000** or less in the **Invoice Amount** index field.

Greater Than or Equal to (>=) Operator

When configuring a search, the **greater than or equal to** (>=) operator is one of the relational operators you can use to specify the relationship between an index field and an index value in your search criteria.

The **greater than or equal to** (>=) operator is located in the **Operator** drop-down list, on the Search page.

> **NOTE** This list is not visible, until you select a document group and document type, and click the **Add condition** button, under the **Search Criteria** heading.

Use the **greater than or equal to** (>=) operator to retrieve only those documents that contain, in the specified index field, an index value that is greater than or equal to the one in the search phrase.

Examples:

Ship Date >= **07/01/2001** retrieves all documents with an index value of **07/01/2001** or later in the **Ship Date** index field.

Invoice Amount >= **15000** retrieves all documents with an index value of **15,000** or more in the **Invoice Amount** index field.

Boolean Operators

Boolean operators enable you to define searches that include more than one search phrase. Searches made up of two or more search phrases can include multiple index fields and index values.

The Boolean operators are located on the Search page.

NOTE This list is not visible, until you select a document group and document type, and click the **Add condition** button under the **Search Criteria** heading.

Click the drop-down arrow to select a Boolean operator from the list.

NOTE While the **AND** operator can be used to link search criteria containing different index fields, the **OR** operator can only be used to link search criteria containing the same index field.

The following Boolean operators are available:

- **AND Operator**: The AND operator specifies that, in order for a document to be retrieved, all search phrases linked by it must be true. The search phrases linked with the AND operator can contain different index fields.

  Examples:

  Ship Date > 01/31/2001 AND Ship Date < 03/02/2001 — This retrieves all documents with ship dates from February 1, 2001 through March 1, 2001.

  Customer name = Betty's Music Store AND Ship Date = 01/31/2001 — This returns only documents for Betty's Music Store that have a ship date of January 31, 2001.

- **OR Operator**: Use the OR operator to configure a search, in which only one of the search phrases linked by it must be true for a document to be retrieved. The OR operator will also retrieve any documents that meet both or all of the search phrases. **NOTE:** The OR operator can only be used to link search criteria containing the same index field. For example, the search phrase Employee Name = John Smith OR Department = Accounting would not work because Employee Name and Department are two different index fields. (This search phrase would work if you linked the search criteria with the AND operator, however.)

  Examples:

  Customer Name = Bluegrass Records OR Customer Name = The Glass Harmonica — This search returns documents for either **Bluegrass Records** or **The Glass Harmonica** (or both, if the document type selected contains documents for both stores).

  Invoice Number = 11187 OR Invoice Number = 11189 — This search will return documents for either invoice **11187** or invoice **11189** or both.

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:

- Storage Management
- Maintenance Management
- Document Management
- Document Retrieval
- Transfer

STORAGE MANAGEMENT

MSAdmin provides a system administrator with tools to define how and where Monarch Server compresses and stores reports. How and where those reports are stored has a direct impact on the overall speed and flexibility of the system.

Reports put into the Monarch Server system are stored in logical storage bins referred to as storage groups. All data in a storage group is stored together, so keep this in mind when establishing these groups. You may want to store all accounting reports in one storage group and all personnel reports in another. All reports having the same retrieval requirements might be stored together. Or, you may want to store all reports in a single storage group. Each organization will have different requirements based on its specific company policies and procedures.

Each storage group may be composed of several storage volumes, each volume having a specific, user-defined size. You may create several copies of each volume, each copy with its own physical location. A single storage group is actually a volume set, and each volume set includes multiple sub-volumes.

Volumes are used as a means of segmenting the increasing number of stored reports in a storage group. Segmenting the number of stored documents into smaller, more manageable pieces facilitates the migration of older data to nearline or offline storage media.

Each volume contains at least one copy. This copy is where the compressed report files that have been processed by Monarch Server are recorded initially. Additional copies may be recorded to other storage locations for data redundancy and long-term archival purposes.

Each storage group contains a set of volumes of the **First Copy** or **Second Copy**. When creating a storage group, the sub-volume size is defined. All the files for the current storage group will be grouped by sub-volumes, each of which will have the fixed maximum size defined by a storage group.

A storage group is also defined by such term as current sub-volume. This is the sub-volume all the new files are created on. If the current sub-volume is completed (or reaches the maximum size), it is closed and a new sub-volume is created. It becomes the current sub-volume. A new sub-volume may be created on any volume with the same type (**First Copy** or **Second Copy**), which is contained in the storage group. The system chooses the first volume, which has enough space for creating another full sub-volume.

For more information about storage groups, see Storage Groups.

For more information about storage volumes, see Storage Volumes.

Storage Volumes

A storage volume is a physical location on a server, to which reports can be archived, and from which documents can be retrieved. You can use storage volumes to segment the continually growing amount of data in a storage group. Segmenting stored data into smaller, more manageable pieces facilitates the migration of older data to nearline or offline storage media.

> **NOTE** Volumes are subdivided into sub-volumes.

Each storage volume in the Monarch Server system is either a **First Copy** or a **Second Copy** volume. When you assign a storage volume to a storage group, you must specify the copy number of the volume (either **1** or **2**).

The first volume you add to the Monarch Server system should be designated a **First Copy** volume. This **First Copy** volume is where the compressed report files that are archived in the system are initially written to. **First Copy** volumes usually reside on fast devices. They should provide quick access because they use the most often retrieved documents. **Second Copy** volumes may reside on slower devices (such as magneto-optical) and can be used for data redundancy and long-term archival purposes.

Each document may reside on the volumes of the **First Copy** or **Second Copy**, or on both. When performing document retrievals, the system attempts to extract data from the volumes of the **First Copy** initially. If the data on the **First Copy** was either moved (Maintenance Task — Move) or copied (Maintenance Task — Copy) into **Second Copy** AND was purged (Maintenance Task — Purge), it will attempt retrieval from the volumes of the **Second Copy**.

See About Copies for additional information.

You can move a document from one copy to another using maintenance tasks.

All storage volumes used in the system are listed on the Volumes page. To display this page, click **Report Warehouse** on the Navigation Tree, then click **Storage** and select **Storage Volumes**. Use this page to add, edit, or remove a storage volume.

To add data to a storage volume

1. Configure a filing task.
2. Run it via a filing process.

To remove data from a storage volume

1. Configure a maintenance task.
2. Run it via a maintenance process.

The volume can be closed to prevent further writing and copying operations to this volume, or can be set as Read Only to restrict access to deletion of this volume.

About Copies

A copy is a storage volume that receives data (reports or documents) when it is archived in the system. Each volume in the Monarch Server system is either a **First Copy** or a **Second Copy** volume. When you assign a storage volume to a storage group, you must specify the copy number of the volume (either **1** or **2**).

The first volume you add to the Monarch Server system should be designated a **First Copy** volume. This **First Copy** volume, which should reside on a mass storage device, is where the compressed report files that are archived in the system are initially written to. Subsequent volumes (that is, **Second Copy** volumes) may reside on any kind of storage medium, and can be used for data redundancy and long-term archival purposes. When performing document retrievals, the data on a **First Copy** volume will be accessed before the data on a **Second Copy** volume.

First Copy Volume

If the data is to be accessed frequently, you may choose to continue to retain the **First Copy** volume until data retrieval requirements subside. As reports are filed to this location, they are maintained in sub-volumes according to the specified sub-volume size. You may specify more than one **First Copy** volume. The first **First Copy** volume in the list will be used until it is full. Then the second **First Copy** volume will be used, etc.

> **NOTE** Data should not be moved or copied to a **Second Copy** volume until you are sure it is "error-free", as data in a **Second Copy** volume cannot be removed from the system.

Second Copy Volume

A **Second Copy** volume is a storage location where reports are archived for long-term storage. Typically, this is an optical storage device (for example, CD-ROMs, CD-ROM drive, etc.). If you don't specify a **Second Copy** volume, data remains in a **First Copy** volume indefinitely.

In practice, the way it works is this: A report is filed into a **First Copy** volume for temporary storage. If there was an error in a report or the archive process, the data can be deleted from the **First Copy** volume. Once the data has been determined to be correct (and/or once the retrieval demand for it decreases enough), it can be copied or moved to a **Second Copy** volume.

> **NOTE** Once data has been moved or copied to a **Second Copy** volume, it cannot be deleted from the system.

There are two ways, via maintenance tasks, to permanently archive data in a **Second Copy** volume:

1. Copy 1 to 2 maintenance task: Copies the data to a **Second Copy** volume, but retains it in a **First Copy** volume.
2. **Move** maintenance task: Moves the data from a **First Copy** volume to a **Second Copy** volume. After that it is possible to delete the **First Copy** volume (**First Copy** volumes can only be deleted if they are empty, that is, if they contain no data). The **Second Copy** volume will be used for all subsequent retrievals.

> **NOTE** If data that has been moved to a **Second Copy** volume needs to be accessed frequently again, the data could be copied back to mass storage for faster retrievals.

Adding a Storage Volume

When you add a storage volume to the system, you designate a physical location on a server for archiving reports. Every time you add a volume, you must assign the volume to a storage group and specify whether it is a **First Copy** or a **Second Copy** volume.

> **NOTE** The first volume you add should be designated as a **First Copy** volume. (See About Copies for more information.)

To add a storage volume

1. On the Navigation Tree, click **Report Warehouse**, then click **Storage** and select **Storage Volumes**.

2. Click + on the **Volumes** page. The **Storage Volume** page appears.

> **NOTE** Field value defaults for some of these fields can be preset in MSAdmin. To do this, click **System Settings** on the Navigation Tree, select **System Preferences** and click the **System Defaults** tab. Setting these defaults can save time and errors in definition of storage volumes.

- Enter a unique name of a new storage volume into the Storage Volume Name field.
- Enter the text defining the storage volume into Description field. This step is optional.
- Select the name of a server from Location list. Select the Any Location option, if you will specify a UNC path in the Path field.
- Select the name of a server from a list. You can select the Any Location option, if you specified a UNC path in the Path field.

> **NOTE** If the required server is not on the list, refer to Locations Management topic to add a new location.

3. **Path**: Enter a path to the volume or use the **Browse** button. This path must be unique. In some cases such as defining a path using UNC paths, it may be required to manually create the path on the server. Typically, Monarch Server will create the storage path if needed.
4. Enter the maximum size of the volume into the **Maximum Volume Size field. Volume s**ize is measured in megabytes or gigabytes depending on the value you select from the list. **Current Size in bytes**: Displays the current volume size when editing only. This value will not be displayed for a new volume.

> **NOTE** The **Location**, **Maximum Volume Size**, and **Path** boxes are available only when adding a new volume.

5. Select **Read only** check box if you want to prevent deletion of a volume.
6. Select **Closed** check box to prevent further writing and copying operations to this volume.

> **NOTE** **Storage Volume is Assigned To|Copy #**: This list contains a storage group with an associated copy number. This is an information box only. The contents of this box cannot be edited.

7. To save a new storage volume, click **Save**.
   If the operation is completed successfully, the system saves the new storage volume to the database and displays the updated **Volumes** page.
8. To clear the recent changes, click **Clear**.
9. To cancel the operation and return to the **Volumes** list without saving data, click **Cancel**. Keep in mind that no changes will be saved.

Assigning Storage Volumes to Storage Groups

After adding a storage volume you can assign it to a storage group. If no storage groups exist, you will need to create one and then add the storage volume to it.

To assign a storage volume to a storage group

1. On the Navigation Tree, click **Report Warehouse**, then click **Storage** and select **Storage Groups**.
2. Click the name of a storage group you want to add a volume to or click anywhere on its row and then click .
3. Drag a storage volume from the Available Storage Volumes pane to the Assigned Storage Volumes|Copy# pane.
4. Under the **Assigned Storage Volumes | Copy #**, select either **#1** or **#2** to specify a copy number for the volume.
5. To assign the volume to the group, click **Save**.

Editing a Storage Volume

To edit a storage volume

1. On the Navigation Tree, click **Report Warehouse**, then click **Storage** and select **Storage Volumes**.
2. On the Volumes page, click the name of the storage volume you want to edit or click anywhere on its row, and then click .
   The **Storage Volume** page appears. For details about how to change its items refer to Adding a Storage Volume.
3. Make the necessary changes and click **Save**.
   The system saves the modified storage volume to the database and displays the updated **Volumes** page.

Deleting a Storage Volume

If you no longer need a storage volume, you can delete it. Before you can delete a volume, however, you must remove all data from it. Volumes that contain any data cannot be deleted. In addition, if the volume you want to delete is the only volume assigned to a storage group, you will not be able to delete it because a storage group cannot exist unless a storage volume is assigned to it.

To delete a storage volume

1. On the Navigation Tree, click **Report Warehouse**, then click **Storage** and select **Storage Volumes**.
2. Click at the end of the row with the storage volume you want to delete. A dialog box will appear.
3. Click **Delete** to delete the storage volume from the list.

> **NOTE** You cannot delete a storage volume, if it contains data files.

Storage Groups

A storage group is a logical storage location for reports filed into the Monarch Server system. Each storage group may include several storage volumes.

All the storage groups used in the system are listed on the Storage Groups page. To display this page, click **Report Warehouse** on the Navigation Tree, then click **Storage** and select **Storage Groups**.

Use **Storage Groups** page to add, edit, or remove a storage group.

The page displays a storage group name, the maximum sub-volume size and the count of existing sub-volumes for each storage group on the list.

Adding a Storage Group

When adding a storage group, you need to specify the maximum size (in megabytes or gigabytes) for any sub-volumes that are subsequently assigned to it. You also need to assign at least one storage volume to the storage group (a storage group must contain at least one volume). If no storage volumes exist, you need to create one before you can add a storage group to the system.

You also need to specify a copy number for the volume. Before specifying this information, you should determine what device type will be used for archiving documents. The media size must be accounted for when defining the sub-volume size.

You should also take into account when the media is to be recorded, and whether it is to be recorded over multiple sessions (requiring a dedicated device) or all at once (for backup).

To add a storage group

1. On the Navigation Tree, click **Report Warehouse**, then click **Storage** and select **Storage Groups**.
2. Click + on the **Storage Groups** page.
3. **Storage Group Name**: Enter a unique name of a new storage group.
4. **Description**: Enter any text defining the storage group. This step is optional.
5. **Maximum Sub-Volume Size**: Enter the maximum size of any sub-volumes measured in megabytes or gigabytes.
   - The size should be no greater than 80% of the media or device size that will serve as the smallest archive storage for the reports contained in this storage group. For example, if your selected archive media is CD-ROM, which has a storage capacity of 650 Megabytes, you would enter 520 Megabytes in this box.
   - E-mail notifications can be configured to alert administrators when a storage volume is running out of space. The notification will be sent when the available free space of a volume is less than 2 x Storage Group size. See System Notifications for additional instructions.

NOTE You cannot change the maximum sub-volume size for an existing storage group.

6. **Force sub-volume promotion**: Select this check box to force sub-volume promotion after a specified number of files. When the number of files in the current (active) sub-volume reaches the specified value, this sub-volume will be forced to close and a new active sub-volume will be created automatically. The number of files can be configured in the Monarch Server properties configuration file. The default value is 10000. It is recommended to select this option to ensure proper performance.
7. **Number of Sub-Volumes**: Displays the number of existing sub-volumes for a storage group. This field is empty for a new storage group.
8. **Available Storage Volumes**: Displays the available storage volumes, created earlier by an administrator. To assign a storage volume to a storage group, drag it from the **Available Storage Volumes** pane to the **Assigned Storage Volumes | Copy#** pane.
9. **Assigned Storage Volumes | Copy#**: Displays the assigned storage volumes with the associated copy number. These storage volumes will be included in the current storage group. Include at least one storage volume of the **First Copy**. You can assign only volumes, whose size is greater or equal to the **Maximum Sub-Volume Size** value. To remove a storage volume from the storage group, drag it to the left pane. To change a copy number for a volume, select either **#1** or **#2**.

10. To save a new storage group, click **Save**.
    If the operation is completed successfully, the system saves the new storage group to the database and displays the updated **Storage Groups** page.
11. To clear the recent changes, click **Clear**.
12. To cancel the operation and return to the **Storage Groups** page without saving data, click **Cancel**. Keep in mind that no changes will be saved.

Assigning Storage Volumes to Storage Groups

After creating a storage volume, you can assign it to a storage group. If there are no storage groups, you need to create one and then add the storage volume to it.

To assign a storage volume to a storage group

1. On the Navigation Tree, click **Report Warehouse**, then click **Storage** and select **Storage Groups**.
2. Click the name of the storage group you want to assign a volume to or click anywhere on its row and then click .
3. Drag a storage volume from the Available Storage Volumes pane to the Assigned Storage Volumes|Copy# pane.
4. Under the **Assigned Storage Volumes | Copy #**, select either **#1** or **#2** to specify a copy number for the volume.
5. To assign the volume to the group, click **Save**.

Editing a Storage Group

To edit a storage group

1. On the Navigation Tree, click **Report Warehouse**, then click **Storage** and select **Storage Groups**.
2. On the Storage Groups page, click the name of the storage group you want to edit or click anywhere on its row, and then click .
   The Storage Group page appears. For details about how to change its items, refer to Adding a Storage Group.
3. You can promote a sub-volume for the current storage group. To do this, select the **Sub-Volume Information** tab, and click **Promote**.

> **NOTE** This tab displays information about the storage group's sub-volumes and is available only for existing storage groups. (For details, see Viewing Information On a Sub-Volume.)

4. Make the necessary changes and click **Save**. The system saves the modified storage group to the database and displays the updated Storage Groups page.

Deleting a Storage Group

To delete a storage group

1. On the Navigation Tree, click **Report Warehouse**, then click **Storage** and select **Storage Groups**.
2. Click at the end of the row with the storage group you want to delete. A dialog box will appear.
3. Click **Delete** to remove the storage group from the list.

Sub-Volumes

A sub-volume, like a storage volume, is a physical location on a server, to which reports can be archived and from which documents can be retrieved. A sub-volume is a subdivision (that is, a subdirectory) of a storage volume.

When you first archive reports into a volume, the reports are automatically stored in the first sub-volume. (Initially, volumes contain only one sub-volume.) If you promote the sub-volume, or if the sub-volume is automatically promoted — for example, when the maximum sub-volume size (specified on the Storage Group page) is reached — the first sub-volume is closed and a second sub-volume is opened. Reports will be archived to this second sub-volume until it is promoted too.

A new sub-volume may be created on any volume of the same type (**First Copy** or **Second Copy**), which is contained in the storage group. The system selects the first volume, which has enough space for creating another full sub-volume.

Volumes can contain an unlimited number of sub-volumes. The only limiting parameter is the volume size (specified on the Storage Volume page when adding a volume). Once the specified maximum volume size is reached, no additional sub-volumes can be opened on that volume.

A storage volume may be closed by an administrator to prevent further writing.

You can view detailed information about the storage group's sub-volumes on the **Sub-Volumes Information** tab.

Promoting a Sub-Volume

A sub-volume is automatically promoted (that is, closed) when it reaches its maximum allowed size. (This maximum size is specified on the Storage Group page, when the storage group is created.) Once a sub-volume is closed, you can no longer archive reports into it. Sub-volumes are numbered sequentially, so each time a sub-volume is promoted, the next sub-volume in the sequence begins. A sub-volume cannot be backed up or removed from the system until it has been promoted.

There may be situations where you would like to force MSAdmin to begin a new sub-volume even though the current one is not completely full. For example, you may want to keep all reports for a particular month on a single sub-volume. Or you may not want to start a new fiscal year on a sub-volume that contains information from the previous year. In such cases, you can manually promote a sub-volume and begin a new one.

To manually promote a sub-volume

1. On the Navigation Tree, click Report Warehouse, then click Storage and select Storage Groups.
2. On the Storage Groups page, click the name of the storage group that contains the sub-volume to be promoted or click anywhere on its row, and then click .
3. Select the **Sub-Volumes Information** tab at the top of the page. This tab lists all sub-volumes of a storage group and provides information about their status (i.e., active or closed), number of files, etc.
4. Click **Promote**. A dialog box appears, asking you to confirm that you want to promote the sub-volume.
5. Click **OK** to promote the sub-volume (or click **Cancel** to cancel the promotion).

Viewing Information on a Sub-Volume

To view detailed information about the sub-volume(s) of the selected storage group

1. On the Navigation Tree, click **Report Warehouse**, then click **Storage** and select **Storage Groups**.
2. Click the name of the storage group that contains the sub-volume or click anywhere on its row, and then click .
3. Select the **Sub-Volumes Information** tab at the top of the page.

   The Volume Information window provides the following information:

- **State**: Displays the current status of the sub-volume, e.g., whether it is active or closed.

> **NOTE** It is not possible to archive data to closed sub-volumes.

- **Number of files**: Displays the number of files in a sub-volume.
- **First Copy volume**: Displays the name(s) of a First Copy volume(s).
- **Size on first copy**: Displays the size (in bytes) of data stored in a First Copy volume.
- **Second copy volume**: Displays the name(s) of a Second Copy volume(s).
- **Size on second copy**: Displays the size (in bytes) of the data stored in a Second Copy volume.
- **Promote**: Click to promote the sub-volume. A dialog box appears. Click OK to promote the sub-volume.
  NOTE: When you promote a sub-volume, you close it. Data cannot be added to closed sub-volumes.
- **Cancel**: Click to cancel the promotion.

MAINTENANCE MANAGEMENT

The topics in this section provide information about:

- Maintenance Tasks
- Log Clearance Tasks
- Maintenance Processes
- Full Text Search

Maintenance Tasks

A maintenance task is the operation of copying, moving, and deleting files to maintain document storage. You need to assign maintenance tasks to a maintenance process in order to execute them.

A maintenance task operates on the document files basis (not document object basis).

All maintenance tasks used in the system are listed on the Maintenance Tasks page. To display this page, click **Report Warehouse** on the Navigation Tree, then click **Maintenance** and select **Maintenance Tasks**.

Use this page to add, edit, or remove a maintenance task from the list.

The following types of maintenance tasks are available on the Maintenance Task page:

- **Block**: Blocks certain reports to deny user access. You can also block reports of a document type by clicking on the Report Instances tab.
- **Change DTA_indices**: Changes indexes for pre-mining models.
- **Copy 1 to 2**: Copies data from First Copy volumes to Second Copy volumes.
- **Copy 2 to 1**: Copies data from Second Copy volumes to First Copy volumes.
- **Delete**: Deletes files from the database and from First Copy volumes. However, the files remain on a Second Copy volume, if any.
- **Drop DTA table(s)**: Removes pre-mined data from the MS database including tables for the reports matching the task criteria.
- **Drop Empty Index Values Tables**: Removes tables with empty index values, to speed up the partition creation. This task is recommended if you update a database created in the previous version of the application.

- **Drop Pre-Mined Data**: Removes pre-mined data from the DTA tables in the MS database for the reports matching the task criteria.
- **Full Text Search**: Searches for reports containing the specified text.
- **Move**: Moves data form First Copy volumes to Second Copy volumes. The reverse moving task (from Second Copy to First Copy) is not supported.
- **Pre-Mined Data**: Pre-extracts table data from reports during a filing process (as part of archiving report data), and puts it into the MS database.
- **Purge**: Files are removed from First Copy volumes only if they exist on Second Copy volumes. The system does not remove files from the database and they are available for viewing and searching as usual.
- **Re-index**: Recreates indexes for documents and is used when one of the Monarch models is changed.
- **Rename**: Renames reports and documents. The name model is set in the Auto-Name and Report Name options of the appropriate Document Type.
- **Restore from Archive**: Restores archived-only reports back into working storage where users can view and mine them.

After configuring a maintenance task, in order to run it, you need to configure a maintenance process and assign the task to it. Then you may either specify the execution frequency for the maintenance process, or execute it manually. When the maintenance process runs, the assigned maintenance task is executed.

Adding a Maintenance Task

With maintenance tasks you can copy, move, delete, purge and re-index data. To run a maintenance task, you create a maintenance process and assign the task to it (or you can assign the task to an already existing process). When the maintenance process is executed, it will run the task(s) assigned to it.

To add a maintenance task

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Tasks**.
2. Click + on the **Maintenance Tasks** page.
3. **Maintenance Task Name**: Enter the name of a new maintenance task.
4. **Description**: Optionally, enter any text defining the maintenance task.
5. **Task Priority**: If there are several tasks in queue, the system will first handle those with higher priority. Select priority from the following options: **Lowest**, **Below Normal**, **Normal**, **Above Normal**, **Highest**.
6. **Task Type**: Specify the type of a task. This list contains the following types:
   - **Block** — Blocks certain reports to deny user access. You can also block reports of a document type by clicking on the Report Instances tab.
   - **Change DTA_indices**: Changes indexes for premining models.
   - **Copy 1 to 2**: Copies data from First Copy volumes to Second Copy volumes.
   - **Copy 2 to 1**: Copies data from Second Copy volumes to First Copy volumes.
   - **Delete**: Deletes files from the database and from First Copy volumes. However, the files remain on a Second Copy volume, if any.
   - **Drop DTA table(s)**: Removes pre-mined data from the MS database including tables for the reports matching the task criteria.
   - **Drop Empty Index Values Tables**: Removes tables with empty index values, to speed up the partition creation. This task is recommended if you update a database created in the previous version of the application.
   - **Drop Pre-Mined Data**: Removes pre-mined data from the DTA tables in the MS database for the reports matching the task criteria.

- **Full Text Search**: Searches for reports containing the specified text.
- **Move**: Moves data form First Copy volumes to Second Copy volumes. The reverse moving task (from Second Copy to First Copy) is not supported.
- **Pre-Mine Data**: Pre-extracts table data from reports during a filing process (as part of archiving report data), and puts it into the MS database.
- **Purge**: Files are removed from First Copy volumes only if they exist on Second Copy volumes. The system does not remove files from the database and they are available for viewing and searching as usual.
- **Re-index**: Recreates indexes for documents and is used when one of the Monarch models is changed.
- **Rename**: Renames reports and documents. The name model is set in the Auto-Name and Report Name options of the appropriate Document Type.
- **Restore from Archive**: Restores archived-only reports back into working storage where users can view and mine them.
- **Search Text**: The following option becomes available if you select the Full Text Search task type. Type here the text you want to find in the report.

7. **Max Result Count (0: unlimited)**: The following option becomes available if you select the **Full Text Search** task type. Limit the number of displayed reports. If you enter **0**, all reports are displayed.
8. **Whole words only** check box: The following option becomes available if you select the **Full Text Search** task type. Select this check box to search for the whole words only.
9. **Case sensitive** check box: The following option becomes available if you select the **Full Text Search** task type. Select this check box to distinguish between upper and lowercase letters while searching.

Criteria section

1. **Based on**: Select one of the following options:
   - **Report/Document Dates**: If you select this option, the current task will process reports/documents with creation dates specified in the Date Range field.
   - **Filing Dates**: If you select this option, the current task will process reports/documents with filing dates specified in the Date Range field.
2. **Date Range**: If you select this option, the current task will process reports with dates specified in the **From ... To** boxes. Click to use the Calendar.
3. **Document Age in days**: If you select this option, the current task will process reports with creation dates specified in the **Younger than ... Older than** boxes.
4. **Except**: If you select this option, the current task will process all reports in the **Document Types** pane with the exception of several latest reports number of which is specified in the **latest** box in conformity with the defined **Based on** parameters, namely **Report/Document Dates** or **Filing Dates**.
5. **Count of Reports/Files**: Specifies that the current task will process reports defined by the first/last count of reports you set in the **First** or **Last** boxes. The reports will be sorted by a document date.
   - **For all selected document types**: applies values from Count of Reports/Files fields to all selected document types.
   - **For each document type selected**: applies values from Count of Reports/Files fields to a selected document type only.
6. **Containing Index Value**: This option becomes available after you select a document group in the section below, and assign document types to the task (provided that index fields of the document type are mapped with the corresponding keyword types). Select this option, if you want to specify an index value as one of the maintenance criteria. When you select it, the **Maintenance Criteria** box to the right becomes available. Click **Maintenance Criteria** to expand a hidden section. Select an index field from the list, select an operator from the list, and type a keyword value. You can lookup available keywords by clicking the **Keyword Lookup** button. Click **Save** and the new maintenance criteria will be displayed in a dimmed box.

Document Types section

Click this header to expand a section with the following items:

1. **Document Groups**: Allows you to select a document group from the list. After selection, the system displays all document types for this group, used by the current task, in the **Document Types** pane.
2. **Document Types**: Displays all document types for the document group, selected from the **Document Groups** list. These are used by the current task.
3. **Filter by**: Enter part of the name of a document type you want to find, and click **Apply**.
4. **Assigned Document Types**: In the **Document Types** pane, select the document types that you wish to assign to the task, and drag them to the **Assigned Document Types** pane.

> **NOTE** The icon allows you to remove assigned document types from the list.

Re-index options

> **NOTE** These options become available if you select the **Re-index** task type.

1. **Perform re-pagination**: Allows to re-paginate the volume data. Becomes available on **Re-index** task selection.
2. **Remove Deleted Documents**: Allows to remove deleted documents during **Re-index** task.
3. **Remove Deleted Documents Only**: Re-Index is not performed, only deleted documents are removed.
4. **Use Index Defaults**: Allows use of default document indexes, including original file name element indexing. This option is only visible when using the **Re-index** Maintenance Task type.
5. **Indexed Fields**: This button appears on **Change DTA_indices** task selection. It calls the **Choose Indexed Fields** dialog box.

Redaction options

> **NOTE** These options become available only if a redaction license is enabled in MS.

1. **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.

> **NOTE** This option becomes available only if you select the **Restore from Archive** task type.

2. **Store mining data redacted**: Select this option to allow pre-mined data to be stored with redaction.

> **NOTE** This option becomes available only if you select the **Pre-Mine Data** task type.

Delete options

> **NOTE** These options become available if you select the **Delete** task type.

1. **Delete Working Reports**: Select this option to delete only working reports.
2. **Delete Report Archives**: Select this option to delete report archives.
3. **Delete All**: Select this option to delete all reports.
4. To save a new maintenance task, click **Save**. If the operation is completed successfully, the system saves the a new maintenance task to the database and displays the updated **Maintenance Tasks** page.
5. To clear the recent changes, click **Clear**.
6. To cancel the operation and return to the **Maintenance Tasks** page without saving data, click **Cancel**. Keep in mind that no changes will be saved.

Copying Data from a First Copy Volume to a Second Copy Volume

Once the demand for documents in a **First Copy** volume decreases sufficiently, you may want to copy or move the data to a **Second Copy** volume.

> **NOTE** Before copying data from a **First Copy** volume to a **Second Copy** volume, make sure that it is error-free, because once it is archived in a **Second Copy** volume, it cannot be deleted from the system.

To copy data from a First Copy volume to a Second Copy volume

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Tasks**.
2. Click + on the Maintenance Tasks page. The **Maintenance Task** page appears.
3. Configure a maintenance task, making sure to select the **Copy 1 to 2** option from the **Task Type** drop-down list.
4. Click **Save**.
5. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Processes**.
6. Click + on the **Maintenance Processes** page.
7. Configure a maintenance process, making sure to assign the maintenance task you just created to it.
8. Schedule the process to run, or run it manually by clicking **Run Now** on the **Maintenance Process** page. When the process is executed, the maintenance task runs and the data is copied from the **First Copy** volume to the **Second Copy** volume.

Copying Data from a Second Copy Volume to a First Copy Volume

Sometimes you need to copy data from a **Second Copy** volume to a **First Copy** volume. You can configure a maintenance task to perform this operation in MSAdmin.

To copy data from Second Copy to First Copy volume

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Tasks**.
2. Click + on the Maintenance Tasks page. The Maintenance Task page appears.

3. Configure a maintenance task, making sure to select the **Copy 2 to 1** option from the **Task Type** drop-down list.
4. Click **Save**.
5. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Processes**.
6. Click + on the Maintenance Processes page. The Maintenance Process page appears.
7. Configure a maintenance process, making sure to assign the maintenance task you just created to it.
8. Schedule the process to run, or run it manually by clicking **Run Now** on the Maintenance Process page.

   When the process is executed, the maintenance task runs and the data is copied from the **Second Copy** volume to the **First Copy** volume.

Moving Data from a First Copy Volume to a Second Copy Volume

You can move data from a **First Copy** volume to a **Second Copy** volume. Before doing so, however, make sure that it is error-free, because once it is archived in a **Second Copy** volume, it cannot be deleted from the system.

To move date from a First Copy volume to a Second Copy volume

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Tasks**.
2. Click + on the Maintenance Tasks page. The Maintenance Task page appears.
3. Configure a maintenance task, making sure to select the **Move** option from the **Task Type** drop-down list.
4. Click **Save**.
5. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Processes**.
6. Click + on the Maintenance Processes page. The Maintenance Process page appears.
7. Configure a maintenance process, making sure to assign the maintenance task you just created to it.
8. Schedule the process to run, or run it manually by clicking **Run Now** on the Maintenance Process page.

   When the process is executed, the maintenance task runs and the data is moved from the **First Copy** volume to the **Second Copy** volume.

Deleting Data from a First Copy Volume

You can delete data from a **First Copy** volume, if the data it contains already resides on a **Second Copy** volume. (A **First Copy** volume cannot be deleted unless the data it contains also resides on a **Second Copy** volume.) If the data doesn't yet reside on a **Second Copy** volume, you must first move or copy it to the **Second Copy** volume.

To delete date from a First Copy volume

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Tasks**.
2. Click + on the Maintenance Tasks page. The Maintenance Task page appears.
3. Configure a maintenance task, making sure to select the **Delete** option from the **Task Type** drop-down list.
4. Click **Save**.
5. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Processes**.
6. Click + on the Maintenance Processes page. The Maintenance Process page appears.
7. Configure a maintenance process, making sure to assign the maintenance task you just created to it.
8. Schedule the process to run, or run it manually by clicking **Run Now** on the Maintenance Process page.

   When the process is executed, the maintenance task runs and the data deleted from the **First Copy** volume.

Purging a First Copy Volume

When you purge a **First Copy** volume, the data it contains is removed. Only **First Copy** volumes can be purged (**Second Copy** volumes cannot be purged), and you can only purge a **First Copy** volume if the data it contains also resides on a **Second Copy** volume. (See Copying Data from a First Copy Volume to a Second Copy Volume for related information.)

> **NOTE** Purging a **First Copy** volume does not remove its files from the database. They are available for viewing and searching as usual.

To purge a First Copy volume

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Tasks**.
2. Click + on the Maintenance Tasks page. The Maintenance Task page appears.
3. Configure a maintenance task, making sure to select the **Purge** option from the **Task Type** drop-down list.
4. Click **Save**.
5. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Processes**.
6. Click + on the Maintenance Processes page. The Maintenance Process page appears.
7. Configure a maintenance process, making sure to assign the maintenance task you just created to it.
8. Schedule the process to run, or run it manually by clicking **Run Now** on the Maintenance Process page.

   When the process is executed, the maintenance task runs and the data purged from the **First Copy** volume.

Re-indexing Archived Documents

Sometimes you need to re-index documents stored in the system (for example, when you change the model or autoname string of a document type). To re-index archived documents, you need to configure and run a re-index maintenance task.

To re-index archived documents

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Tasks**.
2. Click + on the Maintenance Tasks page. The Maintenance Task page appears.
3. Configure a maintenance task, making sure to select the **Re-index** option from the **Task Type** drop-down list.
4. Click **Save**.
5. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Processes**.
6. Click + on the Maintenance Processes page. The Maintenance Process page appears.
7. Configure a maintenance process, making sure to assign the maintenance task you just created to it.
8. Schedule the process to run, or run it manually by clicking **Run Now** on the Maintenance Process page.

   When the process is executed, the maintenance task runs and the documents are re-indexed.

Pre-Mined Data Maintenance Task

The **Pre-Mined Data** maintenance task allows you to pre-extract table data from reports during a filing process (as part of archiving report data), and put it into the MS database, if the document type has a premining model assigned to it.

To pre-mine data

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Tasks**.
2. Click + on the Maintenance Tasks page. The Maintenance Task page appears.
3. Configure a maintenance task, making sure to select the **Pre-Mined Data** option from the **Task Type** drop-down list.
4. Click **Save**.
5. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Processes**.
6. Click + on the Maintenance Processes page. The Maintenance Process page appears.
7. Configure a maintenance process, making sure to assign the maintenance task you just created to it.
8. Schedule the process to run, or run it manually by clicking **Run Now** on the Maintenance Process page.

   When the process is executed, the maintenance task runs and the data is pre-mined.

Dropping Pre-Mined Data

The **Drop Pre-Mined Data** maintenance task allows to remove pre-mined data from the MS database. The archived documents become non pre-mined after running the task.

To drop pre-mined data

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Tasks**.
2. Click + on the Maintenance Tasks page. The Maintenance Task page appears.
3. Configure a maintenance task, making sure to select the **Drop Pre-Mined Data** option from the **Task Type** drop-down list.
4. Click **Save**.
5. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Processes**.
6. Click + on the Maintenance Processes page. The Maintenance Process page appears.
7. Configure a maintenance process, making sure to assign the maintenance task you just created to it.
8. Schedule the process to run, or run it manually by clicking **Run Now** on the Maintenance Process page.

   When the process is executed, the maintenance task runs and the pre-mined data is removed.

Dropping DTA Tables

The **Drop DTA table(s)** maintenance task allows to remove pre-mined data from the MS database including tables for the reports matching the task criteria.

To drop DTA tables

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Tasks**.
2. Click + on the Maintenance Tasks page. The Maintenance Task page appears.

3. Configure a maintenance task, making sure to select the **Drop DTA table(s)** option from the **Task Type** drop-down list.
4. Click **Save**.
5. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Processes**.
6. Click + on the Maintenance Processes page. The Maintenance Process page appears.
7. Configure a maintenance process, making sure to assign the maintenance task you just created to it.
8. Schedule the process to run, or run it manually by clicking **Run Now** on the Maintenance Process page.

   When the process is executed, the maintenance task runs and all premined data including tables is removed from the database.

Drop Empty Index Values Tables

The **Drop Empty Index Values Tables** maintenance task allows to remove tables with empty index values, to speed up the partition creation. This task is recommended if you update a database created in the previous version of the application.

To drop tables with empty index values

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Tasks**.
2. Click + on the Maintenance Tasks page. The Maintenance Task page appears.
3. Configure a maintenance task, making sure to select the **Drop Empty Index Values Tables** option from the **Task Type** drop-down list.
4. Click **Save**.
5. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Processes**.
6. Click + on the Maintenance Processes page. The Maintenance Process page appears.
7. Configure a maintenance process, making sure to assign the maintenance task you just created to it.
8. Schedule the process to run, or run it manually by clicking **Run Now** on the Maintenance Process page.

   When the process is executed, the maintenance task runs and all tables with empty index values are removed from the database.

Changing DTA Indices

The **Change DTA_indices** maintenance task allows to change indexes for a premining model.

To change indices

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Tasks**.
2. Click + on the Maintenance Tasks page. The Maintenance Task page appears.
3. Configure a maintenance task, making sure to select the **Change DTA_indices** option from the **Task Type** drop-down list.
4. Click **Save**.
5. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Processes**.
6. Click + on the Maintenance Processes page. The Maintenance Process page appears.
7. Configure a maintenance process, making sure to assign the maintenance task you just created to it.
8. Schedule the process to run, or run it manually by clicking **Run Now** on the Maintenance Process page.

When the process is executed, the maintenance task runs and the indexes for the pre-mining model are changed.

Full Text Search Task

The **Full Text Search** maintenance task allows to search for reports containing the specified text.

To perform Full Text Search

1. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Tasks**.
2. Click + on the Maintenance Tasks page. The Maintenance Task page appears.
3. Configure a maintenance task, making sure to select the **Full Text Search** option from the **Task Type** drop-down list.
4. Click **Save**.
5. On the Navigation Tree, click **Report Warehouse**, then click **Maintenance** and select **Maintenance Processes**.
6. Click + on the Maintenance Processes page. The Maintenance Process page appears.
7. Configure a maintenance process, making sure to assign the maintenance task you just created to it.
8. Schedule the process to run, or run it manually by clicking **Run Now** on the Maintenance Process page.

   When the process is executed, the maintenance task runs and a link to the reports containing the specified text is displayed in the Full Text Search list.

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** and select **Maintenance Processes**. The Maintenance Processes page appears.
2. On the **Maintenance Processes** page, click the name of the maintenance process you want to add a maintenance task to or click anywhere on its row, and then click .
3. From the **Available Tasks** pane, drag the appropriate 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 Task

When you finish configuring a maintenance task, you can run it to perform the specified maintenance action.

To run a maintenance task

1. Configure a maintenance process.

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

2. Assign the maintenance task to it.
3. Run the maintenance process. When the process is executed, the tasks assigned to it will be run as well.

Editing a Maintenance Task

Maintenance Tasks can be edited via the Maintenance Task page.

To edit a maintenance task

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

Deleting a Maintenance Task

You can delete a maintenance task if you no longer need it.

> **NOTE** If the maintenance task you want to delete is the only task assigned to a process, that process will be deleted as well, since an empty process (that is, a process that contains no tasks) cannot be saved.

To delete a maintenance task

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

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.

Full Text Search

All results of the maintenance full text search type tasks in the system are listed on the Full Text Search page.

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

The list shows:

- **Search Text**: The text you entered while configuring a maintenance task.
- **Search Date**: The date when the search was fulfilled.
- **Task Name**: The name of the maintenance task.
- **Count**: A number of the results found.

- **Whole Words Only**: This field is marked if you selected the Whole Words Only check box while configuring a maintenance task.
- **Case Sensitive**: This field is marked if you selected the Case Sensitive check box while configuring a maintenance task.

You can also:

- Select the users to send notification to, by clicking the icon.
- Delete the result, by clicking the icon.

DOCUMENT MANAGEMENT

When reports are archived in Monarch Server, they are compressed and stored in their entirety. Users retrieving information from the report warehouse, however, may choose to retrieve portions of a report. These portions of reports are called documents. For example, a February music invoice report may contain invoices for dozens of different music stores. A user looking for the February invoice for only Betty's Music Store, however, needs to retrieve just that portion of the file. The invoice for Betty's Music Store can be represented as a single document in the stored February invoice report.

To configure stored information so that users can access it in this way, you need to define how you want to break out documents from the input reports. What makes up a document? In some cases, such as with a file of numerous invoices, the answer is easy: each invoice in the file is a separate document. For other reports, however, you may need input from users to determine how to define documents.

You categorize the documents you define as document types and then assign them to previously defined document groups. For each document type (e.g., Music Orders), you specify index fields (e.g., Customer Name) and then map them to the corresponding indexes in the assigned Monarch model file. You then configure a filing task and execute it via a filing process to archive the documents into the system. (A filing task uses the document type information to parse individual reports and index, compress and store them for later retrieval.) Lastly, you grant users access rights to the document types and security keywords, so that they can specify search criteria to retrieve documents.

The archiving process comprises eight steps, all performed within MSAdmin:

- Define a document group
- Define a document type
- Create index fields
- Map index fields to Monarch indexes
- Specify an auto-name string for the documents
- Define a filing task for the document type
- Schedule a filing process to run the filing task
- Assign user rights to view the document type

NOTE	Though the sequence of these tasks represents a logical progression, you do not have to follow their exact order.

About 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.)

There are three types of model files that can be utilized in Monarch Server:

- **Report Models**: A report model is a model file that contains field, filter, summary and chart definitions (as well as other settings), and which is automatically opened when associated documents are viewed. The primary purpose of the report model is to generate the Data View in MSClient.

> **NOTE** Though you can use a report model for your index model, a report model may contain a number of fields that are irrelevant for searching. These unnecessary fields will only slow down the document retrieval process. So, it would be better to create an index model that contains only the fields that will be used for document retrieval.

- **Index Models**: An index model is a special form of model file that, ideally, should only contain those fields that are to be used as index fields for a given document type. It defines the fields on which you want to index a document, and is used only during the archive process. Index models typically contain fewer fields than report models, thus speeding up the document retrieval process.

> **NOTE** When configuring a document type, you may utilize the same model file for your index and report models. It is not necessary to use a different model file for each of them.

- **Premining Models**: A premining model defines a premining scheme for the filing process (as part of archiving report data), and putting it into the MS database. The primary purpose of a premining model is to generate the Dynamic View in MSClient.

Removing a Model from a Document Type

When removing a model from a document type — whether it be a report, index or premining model — you need to specify a replacement model. (A document type cannot be saved without an assigned model.)

> **NOTE** You will also need to run a **Re-index** maintenance task to update the changes to a document type.

To remove a model from a document type

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types** to display the Document Types page.
2. Click the name of the document type you want to remove a model from or clicking anywhere on its row, and then click .
3. Click the **Index Models** button or the **Premining Models** button to display the **Select Model** page.
4. Select a new model.

5. To add a report model to the document type, drag its name from the **Available Report Models** pane to the **Assigned Report Models** pane.
6. To remove a report model from the document type, drag its name from the **Assigned Report Models** pane to the **Available Report Models** pane.
7. Click **Save** to save the changes.

Index Fields

An index field is a field within a document type that is used as an identifier. Document retrievals are based on these fields of information. For example, an index field for a purchase order could be **Purchase Order Number**, while index fields for employee records might be **Social Security Number** and **Employee Name**.

While an index field is a particular field within a document type, an index value (also called a *keyword value*) is a specific value of an index field. For example, **Q197-4112** could be an index value for a **Purchase Order Number** index field, and **123-45-6789** could be an index value for a **Social Security Number** index field.

Monarch Server uses index fields to locate and retrieve documents according to index values. For example, if a user entered an index value of **123-45-6789** for a **Social Security Number** index field, the index value is used to locate documents for the individual with this social security number.

The first step in creating index fields is to open a report in Monarch and create traps to extract the desired fields of information (for example, invoice numbers, customer names, ship dates, and so on). Then you save the index fields in a model file. (For detailed information on this process, consult the Altair documentation.) When adding a document type to the Monarch Server system, you assign Monarch Server index fields to it and specify which model file you want to associate with it. Then you map the Monarch Server index fields with the ones in the associated model file. Monarch Server users will then be able to use the index fields and values to retrieve documents from the report warehouse.

All indexes used in the system are listed on the Indexes page. To display this page, click **Report Warehouse** on the Navigation Tree, then click **Document Management** and select **Index Fields**.

Use this page to add, edit, remove indexes from the list, or import index fields from existing models.

It is possible to save the index fields' values (index values) in encrypted format using the **Encipher** option.

Also the **Enable index selection** option is available for index fields to enable or disable access to the index values in MSClient application.

Identifying Index Fields

The indexing process begins with the creation of an index model within Monarch, via the same field definition process that you would use to extract fields of data from a report (refer to the Altair documentation for detailed instructions on this process). When creating an index model to be used in Monarch Server Web, however, only those fields that are to be used as index fields need to be identified. When you configure a document type within Monarch Server, you will map these Monarch indexes to the document type's index fields.

Monarch Index Definition Considerations

When creating an index model within Monarch, consider doing the following:

- **Include only the fields you want to index**: The index model file should contain all the fields that you plan to map to index fields for the given document type in Monarch Server. It is not necessary to include other fields in the model — in fact, it is preferable that you do not include them. All fields contained in the model will be extracted from the report as part of the archive process, so unused fields would be extracted unnecessarily. Keeping the number of fields to a minimum will speed up the archiving process.
- **Index fields should be natural fields**: Calculated fields should not be used as index fields unless they are derived from a natural field (that is, a field extracted from your reports).
- **Avoid page header templates**: Index fields cannot be derived from page header templates.

- **Use a detail-level template to access the Data View**: During the archive process, all fields in the index model will be used as index fields, regardless of their template level. You don't need or even necessarily want a detail-level template.

  However, if you do not create a detail template, you will not be able to go to Data View. You may want to go to the Data View in order to gain access to the field properties for the index fields you have defined.

  You can get around this problem in two ways. The first way is to create your lowest-level index field as a detail template, regardless of whether it is truly at the detail level in the report. The second way is to define a detail template temporarily, providing access to the Data View. You can then delete the detail-level model before saving the index model.

- **Match data type and check data length**: As part of the document type configuration process in Monarch Server, index fields are mapped to their corresponding Monarch indexes. In order to ensure proper operation, the data type (and field length, when applicable) of the Monarch Server Web index fields and Monarch indexes must coincide. For example, if in Monarch Server you create a string index field, and then you attempt to map it with a Monarch date index, the mapping will fail due to the incompatibility of their data types.

  In addition, you should check that the defined length of the Monarch indexes is less than or equal to that of the corresponding index fields in Monarch Server Web. If they are longer, Monarch Server Web will truncate the index field's data when it is stored. This may cause errors during the archive process if, for example, an index field is no longer unique due to truncation.

- **Set the Ignore Leading Control Characters setting to zero**: Depending on the source of your report data, your report files may contain one or more extra characters at the beginning of each line. These characters are typically printer control codes that have no meaning to Monarch. Monarch provides a setting that allows you to strip a specific number of characters from your reports when they are viewed so that these characters do not interfere with report data.

  If you supply a number greater than 0 for this setting in your Monarch index model, that number of characters will be stripped from the input report as it is compressed and archived in the warehouse. The characters are permanently removed. If the number is set to 0, no leading control characters are removed from your report. However, you may still choose to ignore those characters when viewing, by setting an appropriate value in the Monarch model file that is launched for viewing.

NOTE	In order to ensure that characters are not stripped from your report files inadvertently, it is recommended that the value be set to 0 for all Monarch index models.

To review or change the Ignore Leading Control Characters setting

1. In Monarch, open the desired report (**File**, **Open Report**) and model files (**File**, **Open Model**), then select **Options**, **Input** (ALT,O,I) from the main menu to display the **Input Options** dialog box.
2. Enter the appropriate number of leading control characters to ignore in the **Ignore n Leading Control Characters** field.

NOTE	It is recommended that you set this number to 0 for the index model.

3. Click **OK** to close the **Input Options** dialog box, then save the change to the model by selecting **File**, **Save Model**.

   - **Turn off the Ignore Form Feed Characters setting**: Many report files use a form feed character to indicate the top of a new page. This is especially useful when breaking reports into documents if the Top of Page option is chosen. Monarch provides a setting that allows you to ignore form feed characters that are embedded in the report. NOTE: For most reports, it is recommended that the setting be unselected in the Monarch index model.

To review or change the Ignore Form Feed Characters setting

1. In Monarch, open the desired report (File, Open Report) and model files (File, Open Model), then select **Options**, **Input** (ALT,O,I) from the main menu to display the **Input Options** dialog.
2. If the **Ignore Form Feed Characters** check box is selected, clear it.
3. Click **OK** to close the **Input Options** dialog, then save the change to the model by selecting **File**, **Save Model**.

Monarch Server Web Index Field Considerations

A few things to consider before adding index fields in Monarch Server:

- **Survey your user's requirements**: The point of an index is to help users find documents in the report warehouse as quickly as possible, so you should be sure to index the fields that users are likely to use when looking up a particular type of document. In many cases, the selection of index fields is obvious. However, that is not always the case. The best way to ensure that you are meeting your user's requirements is to ask them. Take a copy of a representative document and ask users which fields they would want to use to find that document in a database.
- **Choose unique identifiers**: When selecting index fields, it is best to select the most unique identifiers in the document, such as Customer name, Social Security Number, Invoice Number, etc. You should not, in general, select fields that would return a large number of instances, like State, for example. You should also avoid using fields that may not have consistent input. For example, Bill or William in the First Name field could both refer to the same customer, making it difficult to locate all documents for that individual. The more unique the identifier, the more precise the lookup will be. These identifiers are often found as column heads, titles, or page headers. In Monarch terms, they are typically found at an Append level.
- **Index only necessary search fields**: In general, you should avoid indexing line item data. In Monarch terms, this means you should generally not index items that are at the Detail level. These index entries would significantly increase the size of the index and increase the length of time required for the system to search the index tables. In addition, having a lot of index entries means that the initial processing of the report will take longer.

Adding an Index Field

When configuring a document type, one of the steps is to map Monarch Server index fields with Monarch indexes. Before that, you need to configure index fields.

> **NOTE** When configuring an index field in Monarch Server, keep in mind that its data type (and, when applicable, its data length) must be compatible with the data type (and length, if applicable) of the Monarch index you plan to map it to. For more information, see Identifying Index Fields.

To add an index field

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Index Fields**.
2. Click + on the **Index Fields** page. The **Index Field** page appears.
3. Enter a unique name of a new index field in the **Name** field. As the name is only reference identification, an index name may be changed without effect to indexed data.
4. Enter any text defining the index in the **Description** field. This step is optional.
5. Select index field data type:
   - **String**: If you select this option, the String data type is defined. In this case type the number of characters in the Length box to set the maximum length of string fields. It is important to make the field length for string data long enough to accommodate the largest possible value.

- **Date**: If you select this option, the Date data type is defined.
- **Number**: If you select this option, the Number data type is defined. In this case type the number of digits after the decimal point in the Decimals box to set the maximum number of digits after the decimal point.

6. Select **Enable index selection** to enable selection of index values in **Select Index Value** dialog boxes in the MSClient and MSAdmin applications.
7. Select **Enable cross-references table** to be able to use a cross-reference table when index searching for documents.
8. Select **Apply wildcards index field search values on search to replace embedded space(s)** to add the ability to automatically add wildcard(s) to Document Search index search criteria. For example, the client's accounts will show both "XX XXXXX" or "XXXXXXX". The the index field is stored with the spaces but some users enter the spaces and some do not so the system needs to replace the space with * to make it an automatic wildcard.
9. Select **Enciphered** to enable storing index values as an encrypted data. By default, they will not be displayed in **Select Index Value** dialog boxes.

NOTE Extended Security option only.

10. Select **Enable dynamic search** to enable the auto-suggest feature for this index field in MSClient.
11. To save the index field, click the **Save button**.
    The system saves the new index to the database and displays the updated **Index Fields** page, if the operation is completed successfully.
12. To reset the values, click the **Clear** button.
13. To close the **Index Field** page without saving, click **Cancel**. Keep in mind that no changes will be saved.

Editing an Index Field

Index Fields can be edited via the **Index Field** page.

To edit an index in the list

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Index Fields**.
2. On the Index Fields page click the name of the index you want to edit or click anywhere on its row, and then click .
   The **Index Field** page appears. For more information on how to change index options, see Adding an Index Field.
3. To save the changes, click the **Save** button.
   The system saves the modified index to the database and displays the updated **Index Fields** page, if the operation is completed successfully.

Deleting an Index Field

You can delete an index field via the Index Fields page.

To delete an index from the list

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Index Fields**.
2. Click at the end of the row with the index you want to delete. A dialog box appears.
3. Click **Delete** to delete the index from the list.

Importing an Index Field

To import an index field

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Index Fields**.
2. Click the **Import Fields** button on the Index Fields page. The **Select Model** box appears.
3. Select a model you want to use for importing, by clicking its name.
4. The **Import From Model** dialog box appears, displaying the following items:
   - **Import**: Select an index, if you want to add it to the index fields.
   - **Name**: Names of index fields from the selected model.
   - **Type**: The type of each index field (for example, string, date or number).
   - **Size**: The length of each string and number index field.
   - **Encipher**: This option allows to keep index values in encrypted format and hide field values in Select Index Value dialog boxes.
   - **Enable index selection**: The option enables selection of index values in Select Index Value dialog boxes.
   - **Save**: Imports and saves indexes to database.
   - **Cancel**: Closes the window.
   - **Select All**: Selects all indexes.
   - **Select None**: Clears all indexes.
5. Select model fields to import as index fields and click **Save**. If an index field name already exists in the database, the row will appear shaded. The system imports and saves indexes in the database and displays the updated **Index Fields** page with them, if the operation is completed successfully.

Mapping Index Fields to Monarch Indexes

When adding a document type, one of the steps you must perform is to map the assigned index fields to the indexes in the Monarch model. Mapping Monarch Server index fields to indexes in the Monarch model enables you to use the index fields (and their associated index values) to retrieve data from a report.

> **NOTE** Before you can map index fields to Monarch indexes, you need to specify an index model in the **Index Model Name** box on the Document Type page.

To map index fields to Monarch model indexes

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. Either click + to create a new document type, or click the name of a document type that you want to do mapping for.
3. Click the **Mapping** header to expand a hidden section with the following items:
   - **Available Index Fields**: Select an index to assign to the current document type from the list.
   - **Monarch Indexes**: Select a corresponding Monarch index (keyword type) for the current document type.

- **Add To Map**: Click this button to add a selected index field and a corresponding keyword type to the map list. It allows mapping an index field with a corresponding Monarch index (keyword type).
- **Add To Map Hidden**: The same as Add To Map, except that the hidden index field will not be available to users in Simple and Advanced Searches. This feature can be used to add indexes to document types that are used to define security keywords, but are not visible to users.
- **Auto Map**: Click this button to automatically map all available indexes to Monarch indexes (keyword types).
- **Map List**: Displays the list of index fields and corresponding Monarch indexes (keyword types).

NOTE To remove a mapped entry (a pair — an index field with a Monarch index) from the map list, click .

4. Map as many index fields to Monarch indexes as you want to, then click **Save**.

Default Index Values

You can have a default index value automatically assigned to each document in a report file. A default index value is a value you specify for an index field that does not occur within a given report. A default index value, therefore, will not change from document to document in a given report file. Typically, you would use a default index value as a document indicator, much in the same way as a "stamp" for a document. A default index value is automatically assigned to each document during the archive process.

NOTE The use of default index values is optional.

For example, let's say you have 10 report files coming in every month, each one containing the sales information of one of 10 different sales regions. The region number does not appear in the report, but you would like that information to be associated with each document. You could create an index field called **Region Number** and assign it to the document type. As reports for each region are processed, the appropriate value for region number would be assigned to each of the report files as a default index value. All documents generated from this report file will have the default index value assigned to them. You can use default index values as search criteria, as if they were ordinary index values. Thus if you searched for information about Region 3, all the documents in the report file could be part of the search results.

To specify a default index value

1. Create a new index field (for which you'll subsequently specify the default value, see step 3 below).
2. Assign the index field to the appropriate document type
3. Configure a filing task to archive the document type into the report warehouse. When scheduling the filing task, on the **Filing Task** page, click the **Index Defaults** tab at the top of the page to display the **Edit Index Defaults** page, then fill in the boxes.
4. Run the filing task.

Document Groups

A document type group (or simply 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, then, contains one or more document types. A document type is defined by its layout or structure. An invoice, for example, is a document type.

Documents are classified in document groups as a means of organizing information hierarchically and thus speeding up the search process. In other words, when users perform a search, they typically select a document group first, and then select a document type or types from that group to search.

In addition, all document types are assigned to storage groups. The storage group you select during configuration is a default storage group. You can assign document types in the same document group to different storage groups. You would take this approach, for example, if you do not want to treat all the document types in the same way.

Adding a Document Group

When configuring a document type, you must assign it to a document group. So before configuring the first document type, you need to configure a document group.

> **NOTE** Adding a document group is one step in the process of archiving reports.

To add a document group to the list

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Groups**.
2. Click + on the Document Groups page. The Document Group page containing the following items appears:
   - Document Group Name: Enter a unique name of a new document group.
   - Description: Optionally, enter any text to describe the document group.
   - Available Document Types: Contains the list of the document types, from which an administrator selects those to be assigned to the current document group. To add a document type, drag it from the left pane to the right pane.
   - Assigned Document Types: Displays the document types selected by an administrator for the current document group. To remove a document type, drag it from the right pane to the left pane.
   - Filter by: The filters for the Available Document Types list and Assigned Document Types list. Type a document type name (or a part of it), and click Apply to filter the list.
   - Save: Saves a new or existing document group.
   - Clear: Clears all the boxes.
   - Cancel: Cancels the operation.

> **NOTE** If you assign a document type to the current document group, the system automatically excludes it from the document group, which contained it before. If you exclude a document type from the document group, it will not be shown in MSClient.

3. Fill in the boxes of the page and click **Save**. The system saves the new document group to the database.

Editing a Document Group

Document Groups can be edited via the Document Group page.

To edit a Document Group

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Groups**.

2. On the Document Groups page do one of the following:
   - Click the name of the document group you want to change.
   - Select the document group, by clicking anywhere on the row, and then click .

   The Document Group page appears. For details about how to change its items refer to Adding a Document Group.
3. Make the necessary changes and click **Save.** The system saves the modified document group to the database and displays the updated Document Groups page, if the operation is completed successfully.

Deleting a Document Group

You can delete a document group via the Document Groups page.

> **NOTE** Before you can delete a document group, you must remove any document types that are assigned to it.

To delete a document group

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Groups**.
2. Do one of the following:
   - Select the document group you want to delete, by clicking anywhere on the row, and then click .
   - Click at the end of the row with the document group you want to delete.

   A dialog box appears.
3. Click **Delete** to delete the document group from the list.

Document Types

A document type is defined by its layout or structure. An invoice is an example of a document type. All instances of a document type display the same types of information using the same format, though the actual data can vary. For example, Mary's invoice may contain 3 line items, while John's invoice — 11. Both invoices, however, contain index fields for an invoice number, invoice date, and customer name, located in the same relative positions.

All document types used in the system are listed on the Document Types page. To display this page, click **Report Warehouse** on the Navigation Tree, then click **Document Management** and select **Document Types**.

Use this page to add, edit, duplicate, or remove a document type from the list.

Adding a Document Type

Adding a document type is one step in the process of archiving reports.

To add a document type

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. Click on the **Document Types** page. The **Document Type** page appears.
3. **Document Type Name**: Enter a unique name of a new document type.

4. **Description**: Optionally, enter a description of the new document type.
5. **Character encoding**: Select the character set encoding for the current document type (for example, ASCII). If you select the **Default** value then character encoding specified for the index model will be used.

> **NOTE** Changing this value after reports have been filed will not change their input encoding. In order to correctly file reports with an incorrect encoding, you need to delete the existing incorrect reports by a maintenance task, and re-file the original reports into Monarch Server.

6. **Exclude this Document Type from searches**: Select this box to make the document type unavailable for client viewing. Typically, this is used in conjunction with a separate pre-mined document type. The hidden document type will only be used as a data-join source in **Dynamic View**. This is a user-search setting and may be changed without effect to stored documents.
7. **Allow searches by document dates**: In some cases, document dates for user searching may be defined by an index field (e.g. **Report Date** or **Run Date**). In this case the document date fields are redundant and may be hidden. Document dates are still stored and are valid for maintenance operations. This is a user-search setting and may be changed without effect to stored documents.
8. **Request filter before initial Table/Summary view**: In case of reports with a large number of data rows, it may be more efficient to request the user to set view options before the data is displayed. The default action is to display the data before allowing the user to set view parameters. This is a user-search setting and may be changed without effect to stored documents.
9. **Assign preceding values to documents not containing append/page values**: When defining a document type to break the report into logical sections (documents), it is expected that each document will contain at least one case of each defined index field. However, there may be cases where an index value occurs only in the beginning of a report or at certain points that are prior to the document breaking point. In this case, a document can be assigned the preceding value, if it does not contain the value.

> **NOTE** Changing this value will not change indexes for previously filed reports. A **Re-index** maintenance task is required to update indexes for previously stored documents.

10. **Document Group**: Select one document group for the document type to be a member of. Changing this value will associate all reports of this document type to the current document groups assignment.
11. **Storage Group**: Select from the list a storage group for the document type. If you change it, filed reports will remain in the previously defined storage group, while new filings will be stored in the newly defined storage group. Changing a storage group assignment does not affect the ability to search for, or retrieve reports in multiple storage groups.

> **NOTE** Storage groups with no associated storage volumes are not included in the list.

12. **Document State Group**: Select one document state group for the document type. Changing this value will associate all reports of this document type to the current document state group assignment.
13. **Index Model Name**: Specify the name for an index model. Click the **Index Models** button on the right to display the list of existing index models. Specifying an index model is necessary for defining document break conditions.
14. **Premining Model Name**: Specify the name of a premining model. Click the **Premining Models** button on the right to display the list of existing models. Defining a premining model is necessary, if you want to use a premining scheme for the filing process.

NOTE For the model, the field names Def_DocId, Def_FileId, Def_PageNo, and Def_LineNo are reserved for Monarch Server use. If these field names exist in the model already, they will be renamed with the addition of an underscore as a suffix. Alternatively, the field names can be changed by editing the model.

15. **Choose Indexed Fields**: This button allows you to select indexes for a premining model. This is an advanced function included for potential performance reasons only. It is recommended that no pre-mine column indexes be selected initially. This button is enabled only when you add a document type. Changing pre-mine indexes after document type definition is accomplished via Maintenance Tasks/Processes.
16. **Available Report Models**: Contains available report models you can select from.
17. **Assigned Report Models**: Contains report models you select from the available ones. To assign a report model to the document type, drag it from the left pane to the right pane. To de-assign a report model from the document type, drag it to the left pane.

NOTE To break a report into smaller documents, you need to identify a break field for it. The system uses the break field to divide the report into documents. The division is logical only, because the report remains stored as one file. For example, you may choose the **Customer** field as the break field for an invoice report. Therefore, every time the system encounters a change in the **Customer** field (which you may have mapped to a **Customer Name** keyword type), a new document begins. Use the following fields to specify document break fields.

18. **Break At ... line #:** Specify document break fields. You may select one of the following options from the list:
    - Top of current page: If you select this value, the system breaks the report at the top of the page, on which the break value occurs.
    - Break On Line and Line # (Line # being an offset value): If you select this option, Monarch Server breaks the report on the line where the field break value occurs. The Line # field allows you to specify where you want the document breaks to occur in relation to the break field.
      For example, if your report is a list of sales by state, you might want to break out a new document whenever a new state is encountered. To do so, you would choose the Break On Line option, enter a value of 0 (zero) in the Line # field (a value of zero means that there is no offset), select Value Change from the Break When list, then select State from the list of fields. Whenever a change in the state is encountered in the report, a new document is broken out and a new state becomes the first line of that document.

NOTE Negative values are not permitted in the **Line #** field.

19. **Break When:** Select one of the following options from the list:
    - **Always**: Select if you want a new document to begin whenever the break field is encountered, regardless of whether there has been a change in its value. For example, if you specify the Customer break field, and the report contains three invoices for the same customer, then the report will be broken into three documents, even though the customer on each invoice is the same.
    - **Never**: Select if you don't want the report to be broken into separate documents. You might select this option if the report contains a small amount of information that doesn't need to be broken into documents (for example, an invoice report that contains only one invoice).

- **Value Change**: Select if you want a new document to begin whenever the break field value changes. For example, if you specify the Customer break field, and the report contains one invoice for three different customers, then the report will be broken into three documents, one for each customer.

20. **Page header rows**: Optionally, specify in this field the number of page header lines that will be inserted in **Document Search** views, when the Document Break does not occur at the beginning of a printed page. The field is enabled only when **Break On Line** is selected. This setting does not alter the source report and may be changed without effect to filed reports. The display effect is in **Document Search** views only. Set this value to **0**, if the header is not required.
21. **Mapping**: Click this header to expand a hidden section with the following items:
    - **Available Index Fields**: Select an index to assign to the current document type from the list.
    - **Monarch Indexes**: Select a corresponding Monarch index (keyword type) for the current document type.
    - **Add To Map**: Click this button to add a selected index field and a corresponding keyword type to the map list. It allows mapping an index field with a corresponding Monarch index (keyword type).
    - **Add To Map Hidden**: The same as Add To Map, except that the hidden index field will not be available to users in Simple and Advanced Searches. This feature can be used to add indexes to document types that are used to define security keywords, but are not visible to users.
    - **Auto Map**: Click this button to automatically map all available indexes to Monarch indexes (keyword types).

> **NOTE** Before mapping indexes you should select the Index Model and add indexes to the **Assigned Indexes** box on the Document Type page.

22. **Map List**: Displays the list of index fields and corresponding Monarch indexes (keyword types). To remove a mapped entry (a pair — an index field with a Monarch index) from the map list, click .
23. **Auto Name**: Allows you to set document names based on any combination of indexes and document identifiers. Click the **AutoName** button to open the **Auto-Name** dialog box that automatically creates an auto-name format.
24. **Report Name**: Allows you to set report names based on any combination of indexes and document identifiers. Click the **ReportName** button to open the **Report-Name** dialog box that automatically creates a report-name format (in the same way as Auto Name).
25. **Document Date**: Set the document date to display:
    - **Last write time**: Select to use the Last Modified date of file properties as a Document Date.
    - **Index value**: Select to map a date type Monarch field to Document Date. From the enabled list of the date type index fields available in the defined index model, select the field you want to use the date from.

> **NOTE** If the selected date type index field doesn't exist in report, **Last write time** is used as a Document Date. If the selected date type index field contains more than one value, the first field value is used as a Document Date.

26. **Save**: Saves a document type.
27. **Clear**: Clears all the boxes.
28. **Cancel**: Cancels the operation.
29. **Duplicate**: Opens the **Copy Document Type** dialog box, where you can select a document type. All boxes will be filled with values of the selected document type. You can make the necessary changes.
30. Fill in the boxes of the page and click **Save**. The system saves the new document type to the database and displays the updated Document Types page, if the operation is completed successfully.

NOTE You can specify other settings for a document type, including access rights to users, templates, report instances, report security rules, views available in the document search results page, PDF print settings and digital signature for a document type. The corresponding tabs appear on the Document Type page after you save a new document type and open it for editing.

Assigning Document Types to Roles, User Groups and Users

After you configure and save a document type, you can assign it to roles, user groups and users. When you do so, you enable the role members, group members and users to retrieve documents of that type.

When assigning a document type, keep in mind that rights in Monarch Server are cumulative. If, for example, you assign a document type to a role, any user assigned to that role will be able to retrieve documents of that type.

To assign a document type to roles, user groups and users

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**. The **Document Types** page is displayed.
2. Click the name of a document type or click anywhere on its row, and then click .
3. Select the **Rights** tab.

   Important: To make a document type to be available to all users, select **Global** check box and click the **Save** button.
   Otherwise proceed to the next steps.
4. Click **User Groups** to expand a hidden section.
   Select user groups that you want to have access to the document type. To select a user group, drag it from the left pane to the right pane.
5. Click **Roles** to expand a hidden section.
   Select roles that you want to have access to the document type. To select a role, drag it from the left pane to the right pane.
6. In the **Users** section you can select users that you want to have access to the document type.
   To select a user, drag their name from the left pane to the right pane.

NOTE If you don't want a user to retrieve any documents of a particular document type, drag a role, user group, or user from the right pane to the left pane.

7. To save the changes, click the **Save** button.
8. To clear the changes, click the **Clear** button.
9. To close the **Rights** tab without saving, click **Cancel**. Keep in mind that no changes will be saved.

Assigning Document Types to Maintenance Tasks

When adding or editing a maintenance task, you need to specify, which document types you want the task to affect.

To assign a document type to a maintenance task

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Maintenance Tasks** to display the **Maintenance Tasks** page.

2. Create a new maintenance task (by clicking), or select an existing maintenance task (by clicking its name, or by clicking anywhere on the row and then clicking).
3. Click the **Document Types** header to expand a hidden section.
4. In the **Document Groups** list, select a document group that contains the desired document type. All document types assigned to the selected document group appear in the **Document Types** pane.
5. In the **Document Types** pane, choose the document types that you want to assign to the task. To add a document type, drag it to the **Assigned Document Types** pane.
6. Click **Save** to save the changes.

Editing a Document Type

To edit a document type

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. Click the name of the document type you want to change or click anywhere on the row, and then click . The **Document Type** page appears. For details about how to change its items see Adding a Document Type. When you open a saved document type for editing, the following additional tabs become available on the Document type page:
   - **Rights** tab is used to assign the document type to appropriate roles, user groups and users.
   - **Templates** tab is used to manage templates for document types.
   - **Report Instances** tab is used to manage report instances for document types.
   - **Report Security** tab is used to manage report security rules for document types.
   - **Available Views** tab is used to manage views that will be available to a user working with a document type.
   - **PDF Print Settings** tab is used to manage user's print settings for a document type. If not defined, the default system settings will be applied.
   - **Excel Digital Signature** tab is used to add a digital signature to Excel exports. This guarantees that the data provided to a user is not modified.
3. Make the necessary changes and click **Save**. The system saves the modified document type to the database and displays the updated Document Types page, if the operation is completed successfully.

Duplicating a Document Type

When creating a new document type, you can save time by duplicating an existing document type with similar parameters.

To duplicate a document type

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. On the **Document Types** page click .
3. Click the **Duplicate** button at the bottom of the page. The **Copy Document Type** dialog box appears.
4. Click the document type that you want to duplicate. All boxes will be filled with values of the selected document type, except for the **Document Type Name** and **Description**.
5. Enter the name and description (optional) of a new document type, make the necessary changes to other boxes and click **Save**. For details about how to fill in the page items refer to Adding a Document Type.

Deleting a Document Type

To delete a document type

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. Click ✖ at the end of the row with the document type you want to delete.
3. Click **Delete** to delete the document type from the list.

Identifying Document Break Fields

Document break fields are index fields that are used to logically divide reports into documents. As a result, even though entire reports are stored in the warehouse, users are able to retrieve individual documents from within them when performing a search.

Typically, the creator of the original report has set up the document in a way that can help you identify break fields. Reports are generally organized according to some type of sorting criteria. For instance, you might have an employee listing broken out by department within each regional office. Each department could be treated as a separate document. Document breaks do not have to appear on page boundaries. They may occur anywhere within a report file.

You need to identify the criteria used to organize a report and then determine a suitable break field for the report. Here are some specific tips regarding break fields:

- **Imagine the report being cut apart with scissors**. This is the effect of a break field. If a header appears only once on a page and you break that page into more than one document, only the first document from that page will have the header included. You cannot duplicate header information from the report across multiple documents.
- **Look for lines that provide numerical subtotals**. You should be sure to include in a single document all the previous lines that contribute to that subtotal, along with the subtotal line itself.
- **Consider how page boundaries affect your document breaks**. You do not need to be limited by them, but many reports will issue a top-of-page command before beginning a new sort field.
- Break the report into subsections that stand alone and "make sense".
- **Consider how users will be working with the documents**. If users always want to look at an office location in its entirety, then it makes no sense to break out documents by department.
- **Decide how duplicate consecutive break field values should be handled**. If you have a report that contains invoices, for example, you might have 3 or 4 invoices in a row for the same customer. How should that situation be handled? Do your users want to see all invoices for the customer as one single document, or do they want each invoice as a separate document? Either option is available. The user's needs will dictate the appropriate solution.

Once you have identified which field in a report you want to break on, you're ready to break the report into separate documents by specifying a document break field. A document break field creates only virtual breaks in a report. The report is still stored in its entirety, but the document break allows the users to retrieve only portions of the report (documents).

NOTE Specifying a document break field is part of the document type configuration process.

Auto-naming documents

With Monarch Server's auto name feature you can automatically assign names, based on any combination of index fields and document identifiers, to documents filed into the system. It is important to provide meaningful names for documents (that is, names that identify the kind of data the documents contain).

NOTE Before assigning an auto-naming scheme, you need to select an index model and add indexes to the map in the **Mapping** section on the Document Type page.

To auto-name a document

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. Click + to add a new document type, or open an existing document type by clicking its name.
3. Click the **AutoName** button at the bottom of a page to display the Auto Name window.
   The **Auto-Name String** field will display auto name macros after you select them by clicking corresponding buttons at the bottom of the page.
4. Build the auto-name string by clicking any combination of the following buttons:
   - **Space**: Adds a space, a hyphen, and another space (-) between two values in the auto-name string.
   - **Document Type**: Adds the document type macro ([DOCTYPE]) to the auto-name string.
   - **Document Dates**: Adds the document date macro ([DOCDATE]) to the auto-name string.
   - **Document Time**: Adds the document time macro ([DOCTIME]) to the auto-name string.
   - **Date of Filing**: Adds the filing date macro ([FILINGDATE]) to the auto-name string.
   - **Time of Filing**: Adds the filing time macro ([FILINGTIME]) to the auto-name string.
   - **File Name**: Adds the file name macro ([FNE <element number>/<delimeter>]) to the auto-name string. For example, if you specify the element number as 1 and the delimiter as a period (.), then the first element of the file name before the period will be included in the auto-name string.
   - **Index**: Adds an index macro (for example, [INDEX 108.2]) to the auto-name string for the index selected in the Index list.

NOTE Before you click this button, select an index field from the **Index** drop-down list, then enter the number of repeat indexes to add to the auto-name string in the **Repeat** field. To include all values of this index field, enter **A** in the **Repeat** field. In the index macro, the first number is the ID number that was automatically assigned to the index field when it was created. The number (or letter, for example, **A**) following the colon is the number you entered in the **Repeat** field.

5. Enter any static text that you would like to appear in the auto-name string.
6. Click **Save** to save the auto-name string settings and close the Auto Name window.
7. To save the change to the document type, click the **Save** button on the Document Types page.

NOTE A dialog may appear, asking whether you want to rename any documents that have already been stored using this new name. Select Yes to rename existing documents, or No to just use this name for new documents as they are added.

Example of using the auto name feature

To auto name documents of a Checking Statement document type in the following format:

```
Checking Statement - Thomas T. Edwards, 12/11/97
```

Perform the following steps in the **Auto Name** window:

1. Click the Document **Type** button. `[DOCTYPE]` appears in the **Auto-Name String** field.
2. Click the **Space** button. A space, hyphen, space (-) appears in the **Auto-Name String** field after `[DOCTYPE]`.
3. Select **Customer Name** from the **Index** drop-down list, enter **1** in the **Repeat** field, then click the **Index** button. `[INDEX 100.1]` appears in the **Auto-Name String** field after the space, hyphen, space (-).
4. Type a comma (**,**) in the **Auto-Name String** field after `[INDEX 100.1]`.
5. Using the spacebar on your keyboard, enter a space in the **Auto-Name String** field after the comma.
6. Click the **Document Date** button. `[DOCDATE]` appears in the **Auto-Name String** field after the space.

The completed string in the **Auto-Name String** field reads: `[DOCTYPE] - [INDEX 100.1], [DOCDATE]`

Managing Document Type Templates

Monarch Server users can apply templates to documents when exporting them. All document templates used in the system are listed on the Templates page.

> **NOTE** Each template type is available only under the appropriate license.

User can manage document type templates (add, edit, or remove) on the Document Type Templates page.

To display the Templates page

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. Open a document type for editing by clicking its name.
3. Click the **Templates** tab.

Adding a Template to a Document Type

After you assign template to a document type you can assign this document type and template to a Monarch Server user. The user will be able to retrieve documents of this type and apply the specified template to documents when exporting them.

To add a template

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. On the Document Types page, do one of the following:
   - Click the name of the document type you want to add a template to.
   - Select the document type, by clicking anywhere on the row, and then click
3. On the displayed page, select the **Templates**.
4. Click the **Add** icon ( ). The Template page containing the following items is displayed:

- **Name**: The name of a new document template is assigned automatically depending on the selected Template file.
- **Type**: Allows you to select the type of a document template. The type is defined by an application, which will be used for document export.
- **Model**: This field becomes available, if you select ES Style from the Type list. You can select the model type, which is used for extracting data for ES Style. At the customer site, the report model for XSL presentation is optimized for extraction and cannot be used for analysis.
- **Template file**: Select Template file, which will be used for document export.

If you select **ES Style** or **ES Report Style** in the **Type** list, the following options become available:

- **PDF page type**: Select the print size of PDF page from the list.
- **Orientation**: Select PDF page orientation (Portrait or Landscape).
- **PDF page width (inches)**: Specify PDF page width.
- **PDF page height (inches)**: Specify PDF page height.
- **Left margin (inches)**: Specify PDF page left margin.
- **Right margin (inches)**: Specify PDF page right margin.
- **Top margin (inches)**: Specify PDF page top margin.
- **Bottom margin (inches)**: Specify PDF page bottom margin.
- **Server XML/XSL conversion**: Select this checkbox to use server XML/XSL conversion.
- **Enable formatting**: Select this checkbox to enable formatting for templates from user preferences.

5. Fill in the boxes of the page and click **Save**. The system saves the new document template to the database and displays the updated Templates page, if the operation is completed successfully.

> **NOTE** You can assign document type templates to user groups, roles or users via the **Template Rights** tab that is available only for existing templates. For more details see Editing a Template.

Editing a Template of a Document Type

To edit a document template

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. On the Document Types page, do one of the following:
   - Click the name of the document type you want to edit a template for.
   - Select the document type, by clicking anywhere on the row, and then click .
3. On the displayed page, select the **Templates** tab at the top of the page.
4. Do one of the following:
   - Click the name of the template you want to edit.
   - Select the template, by clicking anywhere on the row, and then click .

   The Template page appears. For details about how to change its items see Adding a Template.
5. Make the necessary changes.

6. Click **Save**. The system saves the modified document template to the database and displays the updated Templates page, if the operation is completed successfully.

Deleting a Template from a Document Type

To delete a document template

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. On the Document Types page, do one of the following:
   - Click the name of the document type you want to delete a template from.
   - Select the document type, by clicking anywhere on the row, and then click

   The Document Type page appears.
3. Select the **Templates** tab at the top of the page.
4. Do one of the following:
   - Select the template you want to delete, by clicking anywhere on the row, and then click .
   - Click at the end of the row with the template you want to delete.

   A dialog box appears.
5. Click **Delete** to delete the template from the list.

Report Instances

All reports of the current document type are listed on the Report Instances page.

To display the Report Instances page

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. Open a document type for editing by clicking its name.
3. Click the Report Instances tab.

   The Report Instances page displays the list of reports in the system, including the following information:
   - **Premined** (check marks indicate premined reports)
   - **Blocked** (check marks indicate blocked reports)
   - Report file name
   - Report Date
   - Filing Date
   - **Working** (check marks indicate working reports)
   - **Original** (check marks indicate original reports)

   Use this page to edit a description of existing report, remove it from the list, or block all reports.

Editing a Report Instance

To edit a report description

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. On the Document Types page, do one of the following:
   - Click the name of the document type you want to edit a report for.
   - Select the document type, by clicking anywhere on the row, and then click .

   The Document Type page appears.
3. Select the **Report Instances** tab at the top of the page.
4. Do one of the following:
   - Click the name of the report file you want to edit.
   - Select the report file, by clicking anywhere on the row, and then click .

   The Report Instance page appears.
5. Make the necessary changes (change the date, or select the **Blocked** check box), and click **Save**.

The system saves the modified report description to the database and displays the updated Report Instances page, if the operation is completed successfully.

Deleting a Report Instance

To delete a report

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. On the Document Types page, do one of the following:
   - Click the name of the document type you want to delete a report for.
   - Select the document type by clicking anywhere on the row, and then click .

   The Document Type page appears.
3. Select the **Report Instances** tab at the top of the page.
4. Select the report file by clicking anywhere on the row and then click:
   - — To remove only a working report and unique dependent objects such as index values, annotations, discussion threads, etc.
   - — To remove a working report, an archived original report, and unique dependent objects such as index values, annotations, discussion threads, etc.

   NOTE The associated filing result records are not removed in any case.

   A dialog box appears.
5. Click **Delete** to delete the report from the list.

Blocking Report Instances

You can block all report instances via the **Report Instances** tab, so that reports cannot be searched for or viewed by non-admin users in MSClient.

To block all reports

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. On the Document Types page, do one of the following:
   - Click the name of the document type you want to block reports for.
   - Select the document type by clicking anywhere on the row, and then click .

   The Document Type page appears.
3. Select the **Report Instances** tab at the top of the page.
4. Click to block all reports. Check marks will appear in the **Blocked** column of the table.

> **NOTE** To unblock all reports, click . Users in MSClient will be able to search for and view the reports.

Report Security Rules

MS administrators can apply report security rules to reports when users view them in MSClient. The whole report is divided into two parts: "secure" areas of report which can only be seen by users with explicitly granted access rights (branch level) and "non-secure" areas which can be seen by all users who have access to the document type. Report-level security works in parallel with document-level security (security keywords).

Report Security rules allow sections of reports and report data to be hidden from user's view unless access is allowed. By default, users will not have access to hidden sections of reports unless specifically allowed to access the rule (see Rule Rights).

Rules must be defined before reports are filed. If a rule is defined or redefined after reports are filed, a **Re-index** maintenance task must be performed on all affected reports.

All report security rules used in the system are listed on the Report Security page.

To display the Report Security page

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. Open a document type for editing by clicking its name.
3. Click the **Report Security** tab.

   The **Report Security** tab displays the list of report security rules assigned to a document type. Use this page to add, edit, or remove a rule from the list.

Adding a Report Security Rule

To add a report security rule

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. Select the name of the document type you want to add a rule to by clicking the name of the document type or click anywhere on the row, and then click .
3. Select the **Report Security** tab at the top of the page.

4. Click + to display the **Security Rule** page.
5. Modify the settings accordingly.

   Report Security Rule:

   - **Model name**: Shows the current index model.
   - **Depends on field**: Select a model field from the list.

   Starting Rule: This allows you to define the starting point of the rule being added.

   - **Start At**: This refers to the portion of the report on which the security rule is first applied. The following options are available:
     - **Top of Current Page**: The rule is first applied at the beginning of the current page.

       Example:

       A report contains 50 lines of text. The rule begins on the first line of the report, i.e., the top portion of the page.
     - **On Line**: The rule is first applied on the line specified by the user.
       - **Line #**: This field is enabled only when the On Line option is selected in the Start At list.
       - **On field change**: Specify a Monarch field for triggering the rule.

         Example:

         A report contains 50 lines of text with the following fields available: Customer, Account Number, Contact, Quantity, Description, etc.

         If you wish to hide the Account Number information specifically, set the **On field change** to “Account Number,” and then set **line #** field to 0. The rule is first applied to the field labeled as “Account Number.”

         If you wish to hide, for example, the third line from the field “Contact,” then set the **On field change** to “Contact,” and then set **line #** to 3. The rule is first applied to the third line from the “Contact” field.

         Note that all fields available in the report are listed in the drop-down menu of **On field change**.

   Ending Rule: This allows you to define the end point of the rule being added. The following options are available.

   - **End At**: This refers to the portion of the report on which the security rule is last applied. The following options are available:
     - **End of Current Page**: The rule is last applied at the end of the current page.

       Example:

       In the previous example, you applied the rule to the Account Number field.

       If you wish to hide all suceeding information, set the **On field change** to **End of Current Page**. The report will no longer show information following the Account Number field.
     - **On Line**: The rule is last applied on the line specified by the user.
       - **Line #**: Specify a line offset for the On Line options. This field is enabled only when On Line option is selected in the End At list.
       - **On field change**: Specify a Monarch field for triggering the rule.

         Example:

         If you wish to hide all information provided before a certain field, for example, the Contact field, then set the **On field change** to “Contact,” and then set **line #** field to 0. The rule is last applied to the field labeled as “Contact.”

         If you wish to hide, for example, the third line after the “Contact” field, then set the **On field change** to “Contact,” and then set **line #** to 3. The rule is last applied to the third line from the “Contact” field.

Section Limit: Describe the section limits.

- **Limit section to**: Specify how to limit the section.
  The options are: **No limit**, **Amount of lines** and **End of Current Page**.
  - **Amount of lines**: Specify the amount of lines for the Amount of lines option.
    This field is enabled only when the Amount of lines option is selected in the Limit section to list.

6. To save the new report security rule, click the **Save button**.
   The system saves the new report security rule to the database and displays the updated **Report Security** page, if the operation is completed successfully.
7. To reset the values, click the **Clear** button.
8. To close the **Security Rule** page without saving, click **Cancel**. Keep in mind that no changes will be saved.

Editing a Report Security Rule

To edit a report security rule

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. Do one of the following:
   - Click the name of a document type, to which the rule is assigned.
   - Select the document type, by clicking anywhere on the row, and then click .
3. Select the **Report Security** tab at the top of the page.
4. Do one of the following:
   - Click the name of a rule that you want to change.
   - Select the rule, by clicking anywhere on the row, and then click .

   The Security Rule page appears.
5. Make the necessary changes.
6. To set trustees for the report security rule, click the **Rule Rights** tab.
7. Click the **User Groups** or **Roles** heading to expand hidden sections. To assign a report security rule to a user group, role, or user, drag the appropriate trustees from the left pane to the right pane.

> **NOTE** The Security Rule page has the **Rule Rights** tab only for existing rules.

8. Click **Save**. The Security Rule page appears.
9. Click **Save**. The system saves the modified rule to the database and displays the updated Report Security page, if the operation is completed successfully.

Deleting a Report Security Rule

To delete a rule

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. On the Document Types page do one of the following:

- Click the name of the document type you want to delete a report for.
- Select the document type by clicking anywhere on the row, and then click .

The Document Type page appears.

3. Select the **Report Security** tab at the top of the page.
4. Do one of the following:
   - Select the rule you want to delete, by clicking anywhere on the row, and then click .
   - Click at the end of the row with the rule you want to delete.

   A dialog box appears.
5. Click **Delete** to delete the rule from the list.

Hyperlinks

Values in all character and numeric detail cells in selected Dynamic View columns can have defined hyperlinks to query MES and other systems to display additional information in a new window.

All hyperlinks of the current document type are listed on the Hyperlinks page.

> **NOTE** The **Hyperlink** tab is displayed only if a premining model name is defined for the current document type.

To display the Hyperlink page

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. Open a document type for editing by clicking its name.
3. Click the **Hyperlink** tab.

   The **Hyperlink** tab displays the list of hyperlinks assigned to a document type. Use this page to add, edit, or remove a hyperlink from the list.

Adding a Hyperlink

To add a hyperlink

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. Click the name of the document type you want to add a hyperlink to or click anywhere on the row, and then click .
3. Select the **Hyperlink** tab at the top of the page.
4. Click to display the Hyperlink page.
5. Make changes to the following items:
   a. **Model name**: Shows the current index model.
   b. **Field Name**: Select a field name from the list.
   c. **Hyperlink Type**: Select the hyperlink type:

- **Inner Document**: Select to link to the most recent single Document of the specified Document Type where the Index Value is equal to the cell value from the hyperlink.
- **External Link**: Select a model field from the list.

d. **Connected Document Type**: The option becomes available if you select the **Inner Document** hyperlink type. Select a connected document type to search.

e. **Connected Index Field**: The option becomes available if you select the **Inner Document** hyperlink type. Select a connected index field to use.

f. **Url Template**: The option becomes available if you select the **External Link** hyperlink type. Define a URL template for the link.

g. **Test Value**: The option becomes available if you select the **External Link** hyperlink type. Enter a value you want to test. The field is not required.

h. **Preview**: The option becomes available if you select the **External Link** hyperlink type. Click, to preview the result in a new window.

6. **Save**: Saves the hyperlink. If the operation is finished successfully, the system displays the Document Type page.
7. **Clear**: Clears all the fields.
8. **Cancel**: Cancels the operation.

Editing a Hyperlink

To edit a hyperlink

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. Do one of the following:
   - Click the name of a document type, to which the hyperlink is assigned.
   - Select the document type, by clicking anywhere on the row, and then click .
3. Select the **Hyperlink** tab at the top of the page.
4. Do one of the following:
   - Click the name of a hyperlink that you want to change.
   - Select the hyperlink, by clicking anywhere on the row, and then click .

   The Hyperlink page appears.
5. Make the necessary changes.
6. Click **Save**. The system saves the modified hyperlink to the database and displays the updated Document Type page, if the operation is completed successfully.

Deleting a Hyperlink

To delete a hyperlink

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. On the Document Types page do one of the following:
   - Click the name of the document type you want to delete a hyperlink for.
   - Select the document type by clicking anywhere on the row, and then click .

The Document Type page appears.

3. Select the **Hyperlink** tab at the top of the page.
4. Do one of the following:
   - Select the hyperlink you want to delete, by clicking anywhere on the row, and then click ✖.
   - Click ✖ at the end of the row with the hyperlink you want to delete.

   A dialog box appears.
5. Click **Delete**, to delete the hyperlink from the list.

Available Views

You can select views that will be available to users in the document (report) search results page.

To display the Available Views page

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. Open a document type for editing by clicking its name.
3. Click the **Available Views** tab.

To define data views that are to be available for the current document type, select appropriate check boxes and click **Save**.

PDF Print Settings

You can define PDF export settings for printing from **Report** and **Table/Summary** views. These settings do not affect PDF printing from ES Style views (see Adding a Template).

To display the PDF Print Settings page

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. Open a document type for editing by clicking its name.
3. Click the PDF Print Settings tab.
4. Define the appropriate settings and click **Save**.

Digital Signature Settings

You can set the requirement to provide a digital signature on Excel workbook exports.

Digital signatures can be applied to Excel exports using XSLX format and Monarch XMOD model format only.

A digital signature guarantees that data has not been manipulated after the spreadsheet was created by the Monarch Server.

To display the Digital Signature Settings page

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Document Types**.
2. Open a document type for editing by clicking its name.
3. Click the Digital Signature Settings tab.

   The Digital Signature Settings page contains the following items:

- **Enable Digital Signature on this Document Type**: Select the check box, if you want to enable digital signatures on Excel workbook exports.
- **Certificate Name**: A digital signature may be acquired from a Certificate Authority (CA) or generated by Office 2007 Tools. Monarch Server only supports Digital Signatures based on User Stores (no support for file-based digital signatures). Therefore, the certificate name must always include the container reference ('CN=').
- **Friendly Name**: Optional and will be included with the certificate for the user's information.
- **Require Signature**: Select this check box to force a signature on all exports including those made by users without a user preference of XLSX format.
- **Save**: Click to save the settings.
- **Copy to All Document Types and Save**: Click to copy the settings to all document types and save.
- **Cancel**: Click to cancel the operation.

Filing Tasks

A filing task is used to archive reports into a storage volume. After you have configured a filing task, you can run it via a filing process.

All filing tasks used in the system are listed on the Filing Tasks page. To display this page, click **Report Warehouse** on the Navigation Tree, then click **Document Management** and select **Filing Tasks**.

Use this page to add, edit, or remove a filing task.

Adding a Filing Task

Adding a filing task is one of the steps of archiving reports into the Monarch Server system. (See Archiving Reports for a list of steps you need to perform before you add a filing task.)

To add a filing task

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Filing Tasks**.
2. Click + to display the **Filing Task** page.
3. Fill in the boxes of the page and click **Save**.
   The system saves the new filing task to the database and displays the updated **Filing Tasks** page, if the operation is completed successfully.

The **Filing Task** page contains the following items:

General Options

- **Filing Task Name**: Enter a unique name of a new filing task.
- **Description**: Enter any text defining the filing task. This step is optional.
- **Task Priority**: If there are several tasks in queue, the system will first handle those with higher priority. Select priority from the following options: Lowest, Below Normal, Normal, Above Normal, Highest.

Location

Select the location type from the following options: **Physical Location**, **Virtual Location**.

If you select the **Physical Location** type provide the following information:

- **Location**: Specify the CORBA name of the server, which you can select from the list.
  Select the Any Location option, if you specified a UNC path in the File Path field.

> **NOTE** If the required server is not on the list, refer to Locations Management topic to add a new location.

- **File Path**: Specify the path to a folder containing files to be processed or click the Browse button to locate it.
- **File Name Mask**: Specify an input file mask. You can use wildcard characters * (to substitute for any zero or more characters) and ? (to substitute for a single character).
- **Pin filing task to run only on the specified location**: Enable the check box to specify forcefully a particular machine for filing task implementation. The check box is used if you use distributed MS installation with Datawatch.RemoteServices.FilingDispatcherServiceHost module run on different locations. Enabling the check box displays the Location to run drop-down list.
- **Location to run**: Select a machine from the list for pinning the filing task. If no machine is selected for pinning the filing task, the Monarch Server system will select the most relevant machine for running the filing task. If the machine with stopped Datawatch.RemoteServices.FilingDispatcherServiceHost module is selected, the filing task will not be implemented.

If you select the **Virtual Location** type provide the following information:

- **Virtual Path**: Specify the virtual path to a folder containing files to be processed.
- **File Name Mask**: Specify an input file mask. You can use wildcard characters * (to substitute for any zero or more characters) and ? (to substitute for a single character).

> **NOTE** When editing the page, an administrator may add only one location.
>
> Location Names are case-sensitive. Use only names created during Monarch Server installation.

Document Groups

- **Document Group**: Select a document group from the list to narrow the selection list of document types.
- **Document Type**: Click the Select Document Type button to open a dialog box, where you can select the document type that you want to assign to a task.

Converter Type

Processing Type

- **Read and process internal indexes**. This option is selected by default. It is used for reading text and PDF reports where index values will be extracted and reports will be mined by users.
- **Process as native document**. This option is used for processing any file or document that does not require index values 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, appropriate software should be installed on the client's computer (for example, XLS documents will require MS Excel application or viewer).

File processing during filing

- **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.

File processing after filing

Select one of the options listed below, which describe actions for processing files after filing. Files may be saved, deleted or moved to other directory.

- Leave input files where they are
- Delete input files
- Move input files to the location (specify folder)
- Move files causing errors to folder (specify folder)

> **NOTE** When specifying folder, you may use the naming macros to name the destination folder. To do this, click the **Insert Macro** icon and select the naming macros from the drop-down list.
>
> The **File Path**, **Move input files to the location** and **Move files causing errors to folder** boxes define the path related to the server specified in the **Location** list.

0k files processing before filing

Select one of the options listed below, which describe actions for processing input files with zero size before filing. Files may be deleted or moved to other directory.

- Delete 0k input files
- Move 0k input files to the location (specify folder)

> **NOTE** When specifying folder, you may use the naming macros to name the destination folder. To do this, click the **Insert Macro** icon and select the naming macros from the drop-down list.

File Storage Options

- **Use separate PDF storage in addition to plain text storage**: For filing of PDF reports only. When PDF reports are filed, the text is stripped and stored in Monarch Server for fast and efficient mining operations. If this option is selected, the original PDF report is also stored for separate retrieval.
- **Store Original Report**: When reports are 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, to the storage system.
- **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 Unit can provide dramatically increased storage compression but at a possible cost of reduced retrieval efficiency and performance.

Other Processing Options

- **Automatically start task when a new file appears**: If you select this option, a filing process definition is not required for the filing tasks. New files will be automatically queued and processed.
- **Task priority**: If there are several tasks in queue, the system will first handle those with higher priority. Priority is set by numeric values in inverse relation, meaning the higher the priority — the smaller the number, top priority being "1". By default, if this field is left blank, the system will consider it a top-priority task. (when Auto selected).
- **Log performance statistics**: Select this option to gather the following indexing performance statistics: keywords quantity, documents and pages quantities, time for each indexing stage, total time for task and task status. This information is collected in [Instance_Name][Filing_Task_Name][Date][Time].log files.
- **Block Reports**: Select the check box to block reports, so that reports cannot be searched for or viewed by non-admin users in MSClient.

> **NOTE** You can also block reports via the **Report Instances** tab.

Redaction

Redaction settings are available only if a redaction license is enabled in MS.

- **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.
- **Store mining data redacted**: Select this option to allow pre-mined data to be stored with redaction.
- **Save**: Saves a new or existing filing task.
- **Clear**: Clears all the boxes.
- **Cancel**: Cancels the operation.

> **NOTE** Identical reports may only be filed once. Reports with the same attributes (file name, file date, and file size) as a previous report will not be filed for the same Document Type.

Assigning Filing Tasks to Filing Processes

To execute a filing task, you need to assign it to a filing process. The filing process runs and executes the filing task at a certain time, according to the scheduled execution frequency.

> **NOTE** You can configure a new filing process and add the filing task to it, or you can add the task to an already existing process.

To assign a filing task to a filing process

1. On the Navigation List, click **Report Warehouse**, then click **Filing Processes**. The Filing Processes page appears.
2. Do one of the following:
   - Click the name of the filing process you want to add a filing task to.
   - Select the filing process by clicking anywhere on the row, and then click .

The Filing Process page displays.

3. Drag the task you want to assign to the process, from the **Available Tasks** pane to the **Assigned Tasks** pane.
4. Check the process execution frequency and make any adjustments to it, if necessary.
5. Click **Save**.

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

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

Editing a Filing Task

To edit a filing task

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Filing Tasks**.
2. Do one of the following:
   - Click the name of the filing task you want to edit.
   - Select the filing task, by clicking anywhere on the row, and then click .

   The Filing Task page appears. For details about how to change its items refer to Adding a Filing Task.
3. Make the necessary changes.

   The following tabs become available when you edit an existing filing task:
   - **Pre Processing** tab: Allows you to define the list of applications with appropriate parameters, which will be launched before carrying out the current task. For details, see Extra Processing page.
   - **Index Defaults** tab: Allows you to edit index default values for a filing task. For details, see Edit Index Defaults.
   - **Post Processing** tab: Allows you to define the list of applications with appropriate parameters, which will be launched after carrying out the current task. For details, see Extra Processing page.
   - **Filing Results** tab: Allows you to display the filing results, which are generated for the task where these results are available. For details, see Filing Results page.
4. Click **Save**. The system saves the modified filing task to the database and displays the updated Filing Task page, if the operation is completed successfully.

Deleting a Filing Task

To delete a filing task from the list

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Filing Tasks**.
2. Do one of the following:
   - Select the filing task you want to delete, by clicking anywhere on the row, and then click .
   - Click at the end of the row with the filing task you want to delete.

   A dialog box appears.
3. Click **Delete** to delete the filing task from the list.

Extra Processing Page

The Extra Processing page allows you to define and edit pre-processing and post-processing actions for filing tasks. These actions will be launched before or after carrying out a filing task.

To access the Extra Processing window

1. Click Report Warehouse, then click Document Management and select Filing Tasks.
2. Click the name of a task you want to specify pre-processing or post-processing actions for.
3. Select the **Pre Processing** or **Post Processing** tab at the top of the Filing Task page.

The Extra Processing page contains:

- **Extra Processing** pane: Displays the path to the specified executable file, the specified Ignore Fail setting (that is, Yes or No), and the specified command line parameters.

> **NOTE** Pre- and post-processing actions are executed in the order in which they are listed in the **Extra Processing** pane. To change the order, drag an item up or down.

- The **Add** button: Click to add an extra processing action to the list. The Add Extra Process dialog box appears, containing the following items:
  - **Executable File**: Enter the path to an executable file to be used for pre- or post-processing.
  - **Command Line Parameters**: Enter a command statement and any necessary parameters of a process, which you want to launch before or after carrying out the current task.
  - **Ignore Fail**: Specify the desired Ignore Fail setting by either selecting this check box (to specify a Yes setting) or leaving it unselected (to specify a No setting). The Ignore Fail setting species whether or not the filing task should still be run if a pre- or post-processing action fails. Select Yes to have the filing task run if the pre- or post-processing action fails, or No to prevent the filing task from running if it fails.
- The **Save** button: Saves the new extra process.
- The **Clear** button: Clears all boxes.
- The **Cancel** button: Cancels the operation.

Edit Index Defaults Page

You can specify index defaults for a document type. Whenever documents of that type are archived in the system via a filing task, the default index values are applied to them. Monarch Server users can then retrieve these documents by using the default index values in their search criteria.

To access the Edit Index Defaults page

1. Click Report Warehouse, then click Document Management and select Filing Tasks.
2. Click the name of a task you want to edit index defaults for.
3. Select the **Index Defaults** tab at the top of the page.

The Edit Index Defaults page contains:

- **Default Values** pane: Displays the paired index fields and values you have specified.

NOTE Click to delete an index field and value selected in the **Default Values** pane.

- The **Index** list: Select the index field, for which you want to specify a default value.

NOTE The list includes index fields assigned to the document type specified in the **Document Type** field on the Filing Task page.

- The **Value** field: Enter a default value for the selected index field.

  You can build default values from file-name elements:

  a. Click the **Select** button. The **Index Value** dialog box appears.

  b. Type any optional text in **the Index-Value String** field.

  c. Specify the element number and delimiter.

  d. Click **Apply**. The file name macro ([FNE <element number>/<delimeter>]) will be added to the string. For example, if you specify the element number as **1** and the delimiter as a period (**.**), then the first element of the file name before the period will be included in the index-value string.

  e. Click **Save**.

- The **Add** button: Click to add the paired index field and value to the Default Values field.
- The **Save** button: Saves any changes you have made to the settings.
- The **Clear** button: Clears all boxes.
- The **Cancel** button: Closes the window without saving any changes to the settings.

Filing Results Page

The system collects the following data about all filing task executions:

- Files and documents created by a particular filing task execution
- Filing task start time and date
- Filing task finish time and date

You can view the list of all filing task executions on the Filing Results page.

To access the Filing Results page

1. Click Report Warehouse, then click Document Management and select Filing Tasks.
2. Click the name of the task you want to view filing results for.
3. Select the **Filing Results** tab at the top of the page.

The Filing Results page shows the number of summary documents being processed by all filling tasks during the period indicated in the **From** and **To** columns. The number of summary documents is shown in the **Number of Documents** column. To display the **Start Time** and **Finish Time** of each filing task, click a plus sign in the first column.

Using the drop-down list at the top of the table, you can display filing task executions grouped by the following periods: **None** (all executions are shown), **Daily**, **Weekly**, and **Monthly**.

Click to edit filing task results (change the report date range or block all reports).

An administrator can delete any particular filing task result or a group of filing task results by clicking . When deleting particular groups of filing task results, all selected filing task results will be deleted. When deleting filing tasks results, all files and documents created by the filing task execution will be deleted (similar to maintenance task deletions).

> **NOTE** The number of items shown per page is limited by the value of the **Count of items per page** item in system preference (settings). For more information refer to System Settings.

Using the Naming Macros

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
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.

MACRO	SYMBOL	RESULT
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.
Date Format	&[date.<date increment>.<output format>]	Subtracted date; **<date increment>** could be -1d, +1d, -1m, +1m, -1y, +1y, and **<output format>** could be yyyyMMdd, yyyy-MM-dd, etc. The output format is case-sensitive. For a full list of supported format specifiers, click here.
Input file	&[input]	Input file name
Task name	&[taskname]	Filing task name
Document type	&[doctype]	Document type name

Filing Processes

Filing processes are used to schedule and run filing tasks. When a filing process runs, it executes all filing tasks that are assigned to it.

All filing processes used in the system are listed on the Filing Processes page. To display the Filing Processes page, click **Report Warehouse** on the Navigation Tree, then click **Document Management** and select **Filing Processes**.

The Filing Processes page lists the existing filing 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 Filing Processes page to add, edit or delete a filing 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 filing processes by clicking the icon, below the list.

Adding a Filing Process

To add a filing process

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Filing Processes**.

2. Click + to display the **Filing Process** page.
3. **Name**: Enter a unique name of a new process.
4. **Number of concurrent tasks**: Enter the number of the 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 check box, if you want to suspend the current process.
6. **Available Tasks**: Displays all available tasks, from which you can select the tasks to assign to the current process. To add a task, drag it from the left pane to the right pane.

> **NOTE** You can search for a particular task by typing (part of) its name in the **Filter by** field and clicking **Apply**.

7. **Assigned Tasks**: The tasks in this pane will be assigned to the current process. The tasks are carried out in the order they appear on the list. To change the order, drag an item up or down. To remove a task, drag it to the left pane.
8. **Execution Frequency**: These options allow you to set periodicity of launching the current process. For more information, see Defining Execution Frequency.
9. To save the new process, click the **Save button**.
   The system saves the new process to the database and displays the updated **Filing Processes** page, if the operation is completed successfully.
10. To allow an administrator to launch the process immediately 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**.
11. To reset the values, click the **Clear** button.
12. To close the **Filing Process** page without saving, click **Cancel**. Keep in mind that no changes will be saved.

Editing a Filing Process

To edit a filing process

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Filing Processes**.
2. On the Filing Processes page, do one of the following:
   - Click the name of the filing process you want to edit.
   - Select the filing process, by clicking anywhere on the row, and then click .
3. The Filing Process page appears. For details about how to change its items refer to Adding a Filing Process.
4. Make the necessary changes and click **Save**. The system saves the modified process to the database and displays the updated Filing Processes page.

Deleting a Filing Process

To delete a filing process

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Filing Processes**.
2. Do one of the following:

- Select the filing process by clicking anywhere on the row, and then click ✖.
- Click ✖ at the end of the row with the filing task you want to delete.

A dialog box will appear.

3. Click **Delete** to delete the filing process from the list.

Assigning Filing Tasks to Filing Processes

To execute a filing task, you need to assign it to a filing process. The filing process runs and executes the filing task at a certain time, according to the scheduled execution frequency.

> **NOTE** You can configure a new filing process and add the filing task to it, or you can add the task to an already existing process.

To assign a filing task to a filing process

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Management** and select **Filing Processes**.
2. Do one of the following:
   - Click the name of the filing process you want to add a filing task to.
   - Select the filing process, by clicking anywhere on the row, and then click ✎.
3. Drag the task you want to assign to the process, from the **Available Tasks** pane to the **Assigned Tasks** pane.
4. Check the process execution frequency and make any adjustments to it, if necessary.
5. Click **Save**.

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

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

Running a Filing Process

When a filing process runs, it executes all filing tasks assigned to it. A filing process can be run in two ways:

- **Manually** — To run a filing process manually, click the Run Now button on the Filing Process page. You can do this immediately after you've finished configuring a new process. (NOTE: When you click the Run Now button, there may be a brief delay before the process actually runs. This is due to a time discrepancy between the server's clock and the clock on the local machine. Processes are run according to the time of the server's clock.)
- **Scheduled** — To execute a filing process at a scheduled runtime, specify the execution frequency for it on the Filing Process page, then click the Save button. The process will be executed at the specified time.

DOCUMENT RETRIEVAL

The topics in this section provide information about:

- Search Groups
- Searches
- Search Notifications
- System Notifications
- Filing Notifications
- Offline Tasks
- Offline Processes

Search Groups

Search groups provide a convenient way of grouping similar or related searches together.

All search groups used in the system are listed on the Search Groups page. To display the Search Groups page, click **Report Warehouse** on the Navigation Tree, then click **Document Retrieval** and select **Search Groups**.

Use this page to add, edit or delete a search group.

Adding a Search Group

To add a search group

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Retrieval** and select **Search Groups**.
2. Click + on the **Search Groups** page.
3. **Search Group Name**: Enter a unique name of a new search group.
4. **Description**: Optionally, enter any text defining the search group.
5. **Available Searches**: The searches from this pane may be assigned to the current search group. To add a search, drag it from the left pane to the right pane.

> **NOTE** You can search for a particular query by typing (part of) its name in the **Filter by** field and clicking **Apply**.

6. **Assigned Searches**: Displays the searches selected by an administrator for the current search group. To remove a search, drag it to the left pane.
7. To save a search group, click the **Save button**.
   The system saves the new search group to the database and displays the updated **Search Groups** page, if the operation is completed successfully.
8. To reset the values, click the **Clear** button.
9. To close the **Search Groups** page without saving, click **Cancel**. Keep in mind that no changes will be saved.

> **NOTE** If you assign a search to the current search group, the system automatically excludes it from the search group, which had contained it before. If you exclude a search from the search group, it will not appear in MSClient.

Editing a Search Group

To edit a search group

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Retrieval** and select **Search Groups**.
2. Click the name of the search group you want to edit or click anywhere on its row, and then click . The **Search Group** page appears. For details about how to change its items refer to Adding a Search Group.
3. Make the necessary changes and click **Save**.
   The system saves a modified search group to the database and displays the updated **Search Groups** page, if the operation is completed successfully.

Deleting a Search Group

To delete a search group from the list

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Retrieval** and select **Search Groups**.
2. Click at the end of the row with the search group you want to delete. A dialog box appears.
3. Click **Delete** to delete the search group from the list.

Searches

Searches allow Monarch Server users to query the report warehouse and retrieve documents that match the search criteria. You can add pre-configured searches to the system and then assign them to users (see Adding a Search). You can also grant users the Document Search privilege, which allows them to configure their own searches.

All searches used in the system are listed on the Searches page. To display this page, double-click **Report Warehouse** on the Navigation Tree, then click **Document Retrieval** and select **Searches**.

Use this page to add, edit or delete a search.

Adding a Search

To add a search to the list

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Retrieval** and select **Searches**.
2. Click on the **Searches** page.
3. Search Type: Select either Document Search or Report Search.
4. **Search Name**: Enter a unique name of a new search.
5. **Search Group**: Select a search group, to which the current search will be assigned.
6. **Document Groups**: Select a document group, where the search is to be made.
7. **Available Document Types**: This pane displays available document types, which depends on the selected document group. To add a document type, drag it from the left pane to the right pane.
8. **Assigned Document Types**: This pane displays the assigned document types.
9. **Document Date**: Using the calendar, define the time interval to be covered by the current search.
10. **Annotation Criteria**: Define whether the search should include annotated reports, non-annotated reports or all reports.
11. **Search Criteria**: To specify additional search criteria, including several conditions that involve multiple keyword types and keyword values, click the **Add condition** button.

12. Select a Boolean operator (**AND** or **OR**). While the **AND** operator can be used to link search criteria of different keyword types, the **OR** operator can only be used to link search criteria of the same keyword type.
13. Select a keyword type. The list contains keyword types that are used in the selected document types.
14. Select a relational operator (equal to (=), not equal to (<>), less than (<), greater than (>), less than or equal to (<=), greater than or equal to (>=).
15. Enter a keyword value. To look up available keyword values, click the **Lookup** button.
16. To add another condition, click the **Add condition** button and repeat steps 2-5. To remove a condition, click the **Remove** button.
17. To save a new or existing search, click the **Save button**.
    The system saves the new search to the database and displays the updated **Searches** page, if the operation is completed successfully.
18. To reset the values, click the **Clear** button.
19. To close the **Search** page without saving, click **Cancel**. Keep in mind that no changes will be saved.

Assigning Searches

After you configure a search, you can assign it to roles, user groups and users, so that they can use it.

To assign a search to roles, user groups and users

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Retrieval** and select **Searches**.
2. Select the search you want to assign to roles, user groups, or users:
   - Click the name of the search you want to assign to roles, user groups, or users; or
   - Clicking anywhere on the row, and then click .
3. Click the **Rights** tab at the top of the page. The **Rights** tab is available only for existing objects (in the **Edit** mode).
4. To make a search 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 the section enabling you to select user groups that you want to have access to the search.
   To select a user group, drag it from the left pane to the right pane.
6. Click **Roles** to expand the section enabling you to select roles that you want to have access to the search.
   To select a role, drag it from the left pane to the right pane.
7. In the **Users** section, select users that you want to have access to the search.
   To select a user, drag it from the left pane to the right pane.
8. Click the **Save** button.

Editing a Search

To edit a search

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Retrieval** and select **Searches**.
2. Select the search you want to edit on the **Searches** page:
   - Click the name of the search you want to edit, or
   - Clicking anywhere on the row, and then click .
     The Search page appears. For information on how to change search options, see Adding a Search.

3. To assign the search to roles, user groups and users, select the **Rights** tab at the top of the page.
4. Make the necessary changes and click **Save**. The system saves the modified search in the database and displays the updated Search page, if the operation is completed successfully.

Deleting a Search

You can delete a search from the system at any time. Keep in mind, however, that if you delete the search, users will no longer be able to use it.

> **NOTE** In some cases it makes more sense to edit the search (deny everyone's access rights to it) rather than to delete it outright. If you delete it, in the future if you find that you need the same search (or a similar one), you will have to configure and save it all over again. If, instead of deleting it, however, you remove everyone's access to it, you could easily "activate" it in the future should you need to do so.

To delete a search from the list

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Retrieval** and select **Searches**.
2. Select the search you want to delete on the **Searches** page:
   - Click the name of the search you want to delete, or
   - Clicking anywhere on the row, and then click ✖. A dialog box appears
3. Click **Delete** to delete the search from the list.

Search Notifications

Search notifications are an automated way of notifying Monarch Server users via e-mail that a scheduled search has been run. Only authorized users of the Monarch Server system may receive search notifications.

A scheduled search runs at a certain time according to its specified execution frequency. Search notifications automatically trigger an e-mail to specified users, notifying them that the search has occurred and providing a link to the resulting document list. Users may subscribe to permitted notifications from MSClient.

Unlike system notifications, search notifications are not predefined.

> **NOTE** You can only create, edit and delete notifications if you have been granted **Create Notification** rights by your system administrator. If you have not been granted notification rights, however, and another Monarch Server user with notification rights assigns a notification to you, you can still select it (that is, turn it on) and be notified, when a search occurs.
>
> The creator of the notification automatically receives the rights to the notification.

Search notifications are listed on the Search Notifications page, which you can use to add, edit, or remove search notifications.

To display the Search Notifications page, click **Report Warehouse** on the Navigation Tree, then click **Document Retrieval** and select **Search Notifications**.

Adding a Search Notification

To add a search notification

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Retrieval** and select **Search Notifications**.
2. Click + on the Search Notifications page. The **Search Notification** page appears.
3. **Search Notification Name** box: Enter a unique name for a notification.
4. **Search Groups** drop-down list: Select the search group that contains the search you want to apply a notification to.
5. **Searches** box: Select the search that you want to apply a notification to. (**NOTE:** Available searches are determined by the search group you specified earlier.)

> **NOTE** You can search for a particular query by typing (part of) its name in the **Filter by** field and clicking **Apply.**

6. **Execution Frequency**: To set the periodicity of launching the notification, select an appropriate option and fill in additional fields that appear.
7. To save the new notification, click the **Save** button.
   The system saves the new notification to the database and displays the updated Notifications page, if the operation is completed successfully.
8. To reset the values, click the **Clear** button.
9. To close the **Search Notification** page without saving, click **Cancel**. Keep in mind that no changes will be saved.

> **NOTE** You can assign a notification to a role, user group or user via the **Rights** tab, which is available only for existing search notifications. For more details see Editing a Search Notification.
>
> At a specified time the search runs, and all users, who turned on the notification, receive an e-mail notification.

Editing a Search Notification

You can edit a search notification that you have added. You cannot edit search notifications added by other Monarch Server users. If these notifications have been assigned to you, however, you can turn them on or off (that is, specify whether you want them applied to you or not).

To edit a search notification

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Retrieval** and select **Search Notifications**.
2. Select the search notification you want to edit on the Search Notifications page:
   - Click the name of the search notification, or
   - Select the search notification, by clicking anywhere on the row, and then click .
     The Search Notification page appears. For details about how to change its items refer to Adding a Search Notification.
3. To assign a search to a role, user group or user, click the **Rights** tab at the top of the page.

> **NOTE** The **Rights** tab is available only for existing objects (in the **Edit** mode).

4. To make a search notification available to all users, select **Global** check box and click the **Save** button. Otherwise proceed to the next steps.
5. Expand hidden sections by clicking the **User Groups**, or **Roles** heading. To specify the roles, groups or users that you want to have rights to the notification, drag their names from the left pane to the right pane (in an appropriate section).

> **NOTE** Assigning a notification to a role, user group or user does not guarantee that they will be notified when the search is executed. They will only be notified if they have selected the notification on the Notifications page in MSClient.

6. Make the necessary changes and click **Save**.
   The system saves the modified search notification to the database and displays the updated **Search Notifications** page, if the operation is completed successfully.

Deleting a Search Notification

You can delete any notifications you have created via the Search Notifications page.

> **NOTE** You cannot delete notifications created by other Monarch Server users.

To delete a search notification from the list

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Retrieval** and select **Search Notifications**.
2. Click ✖ at the end of the row with the search notification you want to delete. A dialog box appears.
3. Click **Delete** to delete the search notification from the list.

System Notifications

System notifications are an automated way of notifying Monarch Server users via e-mail that a particular system event has occurred. Only authorized users of the Monarch Server system may receive notifications.

The events that trigger system notification are pre-defined and cannot be edited or deleted. Typically, these types of notifications are sent to system administrators who are monitoring the system. Users may select which of the notifications are sent to them. Notifications may be sent to more than one user.

System notifications are listed on the System Notifications page. They are pre-defined and can not be edited or deleted.

To display the System Notifications page, click **Report Warehouse** on the Navigation Tree, then click **Document Retrieval** and select **System Notifications**.

To assign rights to receive a system notification

1. Click the name of a system notification.
2. To make a system notification available to all users, select **Global** check box and click the **Save** button. Otherwise proceed to the next steps.
3. Expand hidden sections by clicking the **User Groups**, or **Roles** heading. To specify the roles, groups or users that you want to have rights to the system notification, drag their names from the left pane to the right pane (in an appropriate section).
4. Click **Save**.

> **NOTE** Assigning rights to a system notification does not mean that the notification will be sent. It only allows users to receive the notification, if they request it. Each user must specifically request each notification they want to receive.

You can also assign rights to receive system notifications in the following way:

1. In MSAdmin, click **Rights and Privileges**, and select either **Roles**, **User Groups**, or **Users**.
2. Click the name of a role, user group or user, to whom you want to assign rights.
3. Select the **System Notifications** tab at the top of the page.
4. Drag appropriate items from the Available System Notifications pane to the Assigned System Notifications pane.
5. Click **Save**.

Users may receive notifications for the following event types:

- **FILING - Filing process completion events**: Sends an e-mail to the user whenever any filing process completes.
- **FILING - Filing process - Database error events**: Sends an e-mail to the user whenever a database error occurs during a filing process.
- **FILING - Filing process - Generic error events**: Sends an e-mail to the user whenever a non-specific error occurs during a filing process.
- **FILING - Filing process - Disk off-line error events**: Sends an e-mail to the user whenever a disk offline error occurs during a filing process.
- **FILING - Filing process - Not enough space error events**: Sends an e-mail to the user whenever insufficient space error is encountered during a filing process.
- **FILING - Filing process - File system error events**: Sends an e-mail to the user whenever a file system error is encountered during a filing process.
- **FILING - Filing process - No sub volume error events**: Sends an e-mail to the user whenever no sub-volume error is encountered during a filing process.
- **MAINTENANCE - Maintenance process completion events**: Sends an e-mail to the user whenever any maintenance process completes.
- **DISK_OFFLINE - Offline disk request events**: Sends an e-mail to the user whenever any offline disk request is issued.
- **VOLUME_FULL - Volume full events**: Sends an e-mail to the user whenever a volume becomes full.
- **VOLUME_ALMOST_FULL - Volume almost full events**: Sends an e-mail to the user whenever the sum of the used volume size and twice the amount of the maximum size of the subvolume is greater than or equal to the maximum size of the volume, i.e., used volume size + (max subvolume size) × 2 >= max volume size. This condition is checked before subvolumes rollover in volume (the first subvolume is created automatically).
- **LAST_VOLUME - Last volume usage events**: Sends an e-mail to the user whenever documents begin to be archived in the last volume in a storage group.

- **ONLY_ARCHIVE_REPORT - Working report request events**: Sends an e-mail to the user whenever a request is submitted to restore an archived report to a working state.
- **FILE_OFFLINE - Offline file request events**: Sends an e-mail to the user whenever an offline file request is made.
- **ERROR - All error events**: Sends an e-mail to the user whenever any process results in an error condition.
- **WARNING - All warning events**: Sends an e-mail to the user whenever any process receives a warning.
- **SEARCH - New search creation events**: Sends an e-mail to the user whenever a new search is created.

Filing Notifications

Filing notifications are an automated way of notifying Monarch Server users via e-mail that a filing event has occurred. Only authorized users of the Monarch Server system may receive filing notifications.

Filing notifications are listed on the Filing Notifications page, which you can use to add, edit, or remove notifications.

To display the Filing Notifications page, click **Report Warehouse** on the Navigation Tree, then click **Document Retrieval** and select **Filing Notifications**.

> **NOTE** Users may subscribe to or unsubscribe from filing notifications via the MSClient application.

Adding a Filing Notification

To add a filing notification

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Retrieval** and select **Filing Notifications**.
2. Click + on the Filing Notifications page. The **Filing Notification** page appears.
3. **Filing Notification Name**: Enter a unique name of a new notification.
4. **Filing Description**:Optionally, enter any text defining the filing notification
5. **Document Groups**: Select a document group that contains the document type you want to apply the notification to.
6. **Document Types**: Select the document types to be included in the filing notification definition.
7. **Run immediate**: Select this check box to execute the filing notification immediately upon the filing completion event. Each filing notification will be sent as an individual e-mail.
8. **Execution Frequency**: These options become available if you have not selected the **Run immediate** check box. You can set the frequency of executing filing notifications.
9. To save a new or existing filing notification definition, click the **Save** button.
10. To reset the values, click the **Clear** button.
11. **Save**: Saves a new or existing filing notification definition.
12. To close the **Filing Notification** page without saving, click **Cancel**. Keep in mind that no changes will be saved.

> **NOTE** You can assign a notification to a role, user group or user via the **Rights** tab, which is available only for existing search notifications. For more details see Editing a Filing

Notification.

Editing a Filing Notification

You can edit any filing notification that you have added. You cannot edit filing notifications added by other Monarch Server users. If these filing notifications have been assigned to you, however, you can turn them on or off (that is, specify whether or not you want them applied to you) in MSClient.

To edit a filing notification

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Retrieval** and select **Filing Notifications**.
2. Select the filing notification you want to edit on the **Filing Notifications** page:
   - Click the name of the filing notification you want to edit, or
   - Click anywhere on the row, and then click .
     The Filing Notification page appears. For details about how to change its items, see Adding a Filing Notification.
3. To assign a filing notification to a role, user group or user, click the **Rights** tab at the top of the page.

> **NOTE** The **Rights** tab is available only for existing objects (in the **Edit** mode).

4. To make a filing notification available to all users, select **Global** check box and click the **Save** button. Otherwise proceed to the next steps.
5. Expand hidden sections by clicking the **User Groups**, or **Roles** heading. To specify the roles, groups or users that you want to have rights to the notification, drag their names from the left pane to the right pane (in an appropriate section).

> **NOTE** Assigning a notification to a role, user group or user does not guarantee that they will be notified when a filing task is executed. They will only be notified if they have selected the notification on the Notifications page in MSClient.

6. Make the necessary changes and click **Save**.
   The system saves the modified filing notification to the database and displays the updated **Filing Notifications** page, if the operation is completed successfully.

Deleting a Filing Notification

You can delete any filing notifications you have created via the Filing Notifications page. (You cannot delete notifications created by other Monarch Server users.)

To delete a filing notification

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Retrieval** and select **Filing Notifications**.
2. Click at the end of the row with the filing notification you want to delete. A dialog box appears.
3. Click **Delete** to delete the filing notification from the list.

Offline Tasks

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 of a size exceeding the system limit, Altair 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.

All offline tasks used in the system are listed on the Offline Tasks page. To display the page, click **Report Warehouse** on the Navigation Tree, then click **Document Retrieval** and select **Offline Tasks**.

The **Offline Tasks** page displays the following information:

- **User Name**: Displays the user name for an offline task.
- **Task Name**: Displays the name of an offline task.
- **View Type**: Displays one of the following view types: Table, Summary, Excel, or PRF.
- **Creation Date**: Displays the creation date and time of an offline task.
- **State**: Displays the state of the offline task. It can be one of the following: Ready, Processing or Pending.

To delete an offline task from the list

1. Click ✖ at the end of the row with the offline task you want to delete. A dialog box appears.
2. Click **Delete** to delete the offline task from the list.

Offline Processes

Offline processes are used to schedule and run offline tasks. When an offline process runs, it executes all offline tasks that are assigned to it.

All offline processes used in the system are listed on the Offline Processes page. To display the Offline Processes page, click **Report Warehouse** on the Navigation Tree, then click **Document Retrieval** and select **Offline 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 **Offline Processes** page to add, edit, or remove an offline process from the list.

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.

Adding an Offline Process

To add a process

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Retrieval** and select **Offline Processes**.
2. Click + on the Offline Processes page. The Offline Process page appears.
3. Fill in the boxes of the page and click **Save**. The system saves the new process in the database and displays the updated Offline Processes page, if the operation is completed successfully.

This page contains the following items:

- **Name**: Enter a unique name of a new process.

> **NOTE** All tasks with the **Pending** status will be automatically assigned to the current process.

- **Processing Length**: Specify the value between 1 and 60 for the field processing length.
- **Suspended**: Select this check box to suspend the current process.
- **Execution Frequency**: These options allow you to set periodicity of launching the current process. For more information, see Execution Frequency.
- **Save**: Saves a new process.
- **Run Now**: Allows an administrator to launch the process immediately and saves the schedule in the database with the Run Once scheduling type and the current date as a Date to run.
- **Clear**: Clears all the boxes.
- **Cancel**: Cancels the operation.

Running an Offline Process

When an offline process runs, it executes all offline tasks assigned to it. A offline process can be run in two ways:

- **Manually** — To run an offline process manually, click the Run Now button on the Offline Process page. You can do this immediately after you've finished configuring a new process, or you can manually run a previously configured process by going to the Offline Processes page, selecting the process you want to run, and then clicking the Run Now button.

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

- **Scheduled** — To execute an offline process via a scheduled runtime, specify the execution frequency for it on the Offline Process page, then click Save. At the specified time, the process will be executed.

Editing an Offline Process

To edit an offline process

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Retrieval** and select **Offline Processes**.

2. Do one of the following:
   - Click the name of the offline process you want to edit.
   - Select the offline process, by clicking anywhere on the row, and then click .

   The Offline Process page appears. For details about how to change its items refer to Adding an Offline Process.
3. Make the necessary changes and click **Save**. The system saves the modified offline process to the database and displays the updated Offline Processes page, if the operation is completed successfully.

Deleting an Offline Process

To delete an offline process

1. On the Navigation Tree, click **Report Warehouse**, then click **Document Retrieval** and select **Offline Processes**.
2. Do one of the following:
   - Select the offline process you want to delete, by clicking anywhere on the row, and then click .
   - Click at the end of the row with the offline process you want to delete.

   A dialog box appears.
3. Click **Delete** to remove the offline process from the list.

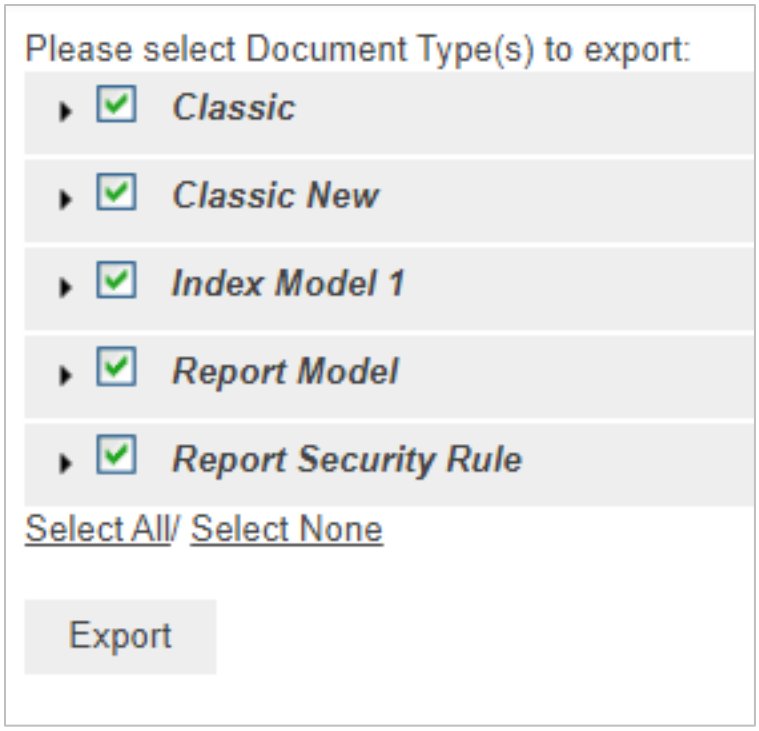
TRANSFER

The topics in this section provide information about importing and exporting of document types, document types configurations and objects related to them.

Export

To export document types

1. On the Navigation Tree, click **Report Warehouse**, then click **Transfer** and select **Export**.
2. Select the document types you want to export. Multiple selection is available.



3. Under each document type you can specify items for export.

> **NOTE** The following items are exported by default: Index Model, Document Type Group, Index Fields, PDF Settings, Digital Signatures, and Available Views.

4. Click the **Export** button and save the exported file.

> **NOTE** The output of the export operation will be an XML file with the following name *Migration_MM_DD_YY_hh_mm_ss PM.xml.*

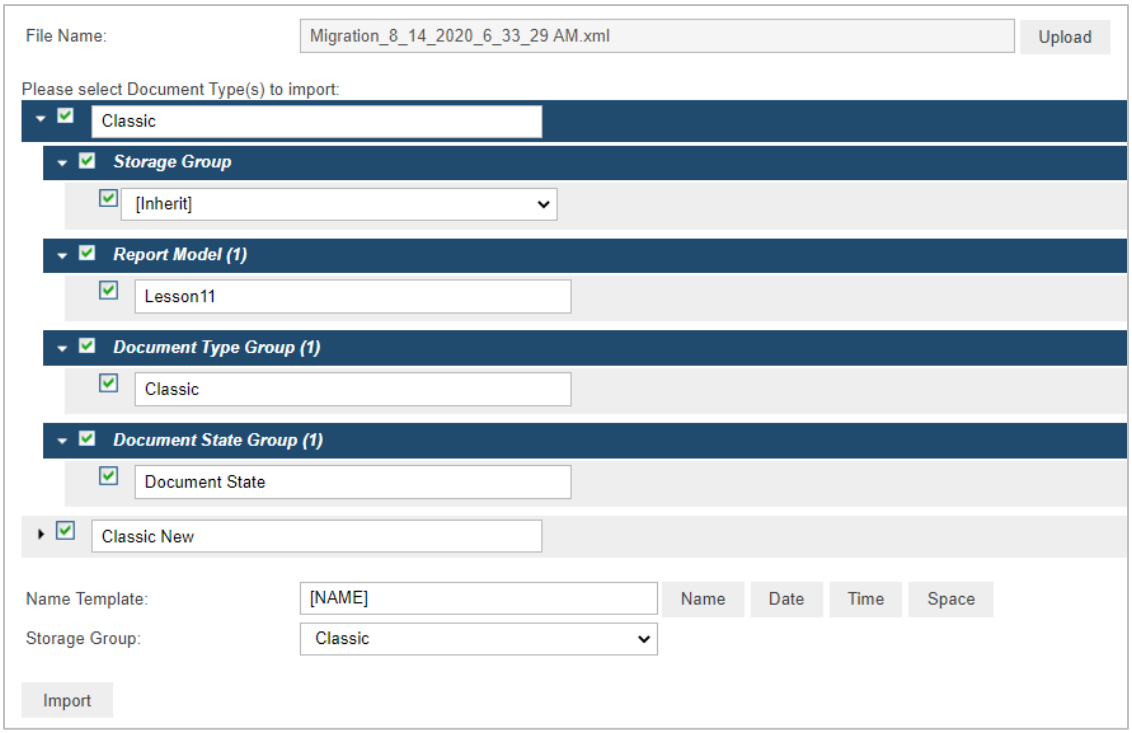
Import

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

To import document types

1. On the Navigation Tree, click **Report Warehouse**, then click **Transfer** and select **Import**.
2. On the displayed page, click the Upload button ans select the .xml file you want to import.

NOTE You can only import an output of a previous export document type operation. The output is an XML file with the following name *Migration_MM_DD_YY_hh_mm_ss PM.xml.*



3. Select document type to import. Multiple selection is available.
4. Under each document type you can specify items for export.

NOTE The following items are imported by default: Index Model, Document Type Group, Index Fields, PDF Settings, Digital Signatures, and Available Views.

5. Under **Storage Group**, select the target storage group from the list. If you select the *[inherit]* value then the imported document type(s) will be placed to the storage group specified in the **Storage Group** field at the bottom of the page.
6. In the **Name Template** field, specify the name(s) for the imported file(s). For this either type in the name you want to assign to the imported document type(s) or insert the following values: name, date, time or space, by clicking the appropriate buttons.

NOTE The name specified in the **Name Template** field will be used for all imported document types. You can also specify names for each item in the editable fields.

7. Click the **Import** button to import document type(s).

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 it's 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 proc^{es}s on the 6th business day of the month, enter “6” in the field.

      To run the proc^{es}s on the 1st business day of the month, then aga^{in} 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. Fom 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/excelexports.

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 ✖ 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 jog 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

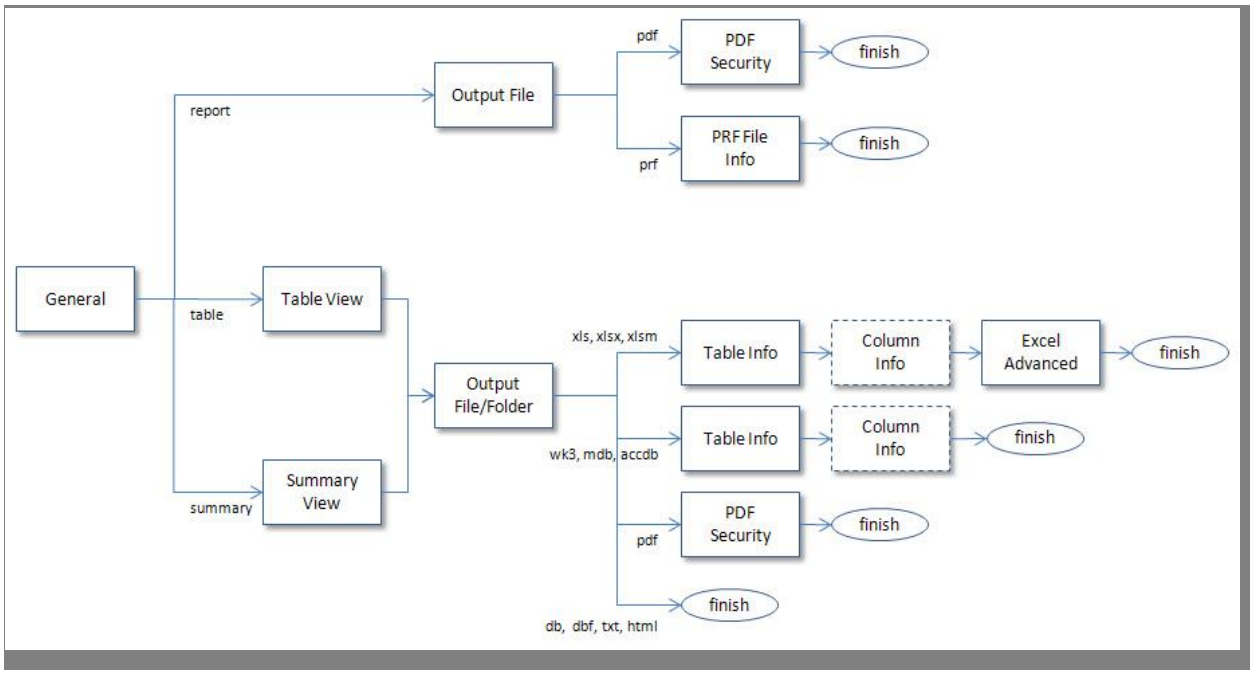
> **NOTE** A set of screens you will traverse depends on your settings. Refer to the export wizard workflow below.

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: ❑ **Save new process**: Save the specified work flow as a new process. ❑ **Save as template**: Save the specified work flow as a template. Elements settings are not saved. ❑ **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: ❑ Schedules ❑ Inputs Monitoring

BUTTON NAME	DESCRIPTION
	❑ Manifests Monitoring ❑ Report Warehouse Only one of them can be chosen at a time. **SCHEDULES TRIGGER** 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. **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 add a new schedule for a process** 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. **To delete a schedule** 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. Select the **Suspend** check box to temporarily hold up the trigger. **INPUTS MONITORING TRIGGER** 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. **NOTE**: Changes in monitoring settings, including enabling and disabling monitoring or changing paths for monitored folders, take effect after approximately 1 minute. **To specify an inputs monitoring file** 1. Click the **Triggers** icon. 2. On the **Process triggers** dialog, select the **Inputs Monitoring** radio

BUTTON NAME	DESCRIPTION
	button. 3. 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. 4. To add a file not from the process, expand the **Browse** section, set the file path, and click the **Add** button. **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. 5. Expand **Options** to specify 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. 6. Click the **Ok** button to save the process monitoring. **To delete an inputs monitoring file** 1. Click the **Triggers** icon. 2. Select the **Inputs Monitoring** radio button. 3. Click **Delete** icon near the file path you want to delete. 4. Click the **Ok** button to save process monitoring. **MANIFESTS MONITORING TRIGGER** 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. **To specify a manifests monitoring file** 1. Click the **Triggers** icon. 2. On the **Process** triggers dialog, select the **Manifests Monitoring**

BUTTON NAME	DESCRIPTION
	radio button. 3. To add a file, expand the **Browse** section, set the file path, and click the **Add** button. **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.) 4. Click the **Ok** button to save the process monitoring. **To delete a manifests monitoring file** 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. **REPORT WAREHOUSE TRIGGER** This trigger is available only if in the Monarch Server License the Enterprise Server component is enabled. 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. **To assign Report Warehouse document types** 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**. You can also add document types from the **Report Warehouse Input** of the current process by clicking the **Assign from process** button. Select the **Disable Triggers** check box to temporarily turn off all triggers.
Scripts	Click to apply scripts to the current process. In the **Process Declarations** tab, the following settings are available. ❑ **Script field**: Displays the existing process declarations. ❑ **Insert Script**: Allows you to quickly generate VB.NET script for frequently used tasks. For complete options, 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. In the **References and Imports** tab, specify additional assembly references and namespaces that are required by PostExport, PreExport,

BUTTON NAME	DESCRIPTION
	and Global Declarations sections. The following settings are available: ❑ **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.
Generate Manifest File	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. In the **Manifest** window, select the preferred manifest file format: ❑ **XML** – creates a manifest XML file ❑ **JSON** – creates a manifest JSON file **NOTE:** This icon is available only after the visual process is saved. **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.
Alerts	Click to view, add, edit, or remove alerts for the current process. 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: ❑ **Code:** Specify the alert type. ❑ **Description:** Specify the alert description. ❑ **Ignore Global Alert:** Select this check box to ignore global alerts. **NOTE:** This option is available only when the **Allow ignore Global alerts** setting is enabled. See Automator Settings for details. ❑ Immediate alert: Select the Raise the alert immediately when detected within running process check box if the alert being created should be immediate. **NOTE:** The **Rise Alert Immediate** checkbox is available only for the SystemAlertNoInputToExporter alert and the SystemAlertProcessElementFailed alert. ❑ **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	Click to open the Log Mode view. 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. 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. **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 **Designer Mode** icon to leave the Log Mode view.
Process Global Settings	Click to modify the workspace validation settings of the Visual Process in the **Process Global Settings** window. Select the **Disable Validation** checkbox to turn off the workspace validation for the loaded Visual Process. **NOTE:** This setting has higher priority than the workspace validation setting in Automator General Settings.
Rights	Click to assign process access rights to other users, roles, or groups. **NOTE:** See Application Settings for more details.
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	❑ **Name** — name of the graphic element. ❑ **Input Path** — path that shows the source file location. The **Input Path** parameter can be filled out manually or using **Browse**. Actions: 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
		• **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). **NOTE:** Input files in compressed archive format (e.g., zip and gzip) can be used and specified. 4. Click the **OK** button.
Web Input	Text file located in the web (FTP/SFTP, HTTP location, HTTPS, Amazon S3)	❑ **Name** — name of the graphic element. ❑ **Input Address** — URL of the source file. 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.) **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.
Database Input	Database file	❑ **Name** — name of the graphic element. ❑ **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: • Local or Web File • Data Link Wizard • Data Link File • Connection Definitions Then select the required file using the displayed dialog. **NOTE:** In addition to Excel file types .xls and .xlsx, .xlsb and .xlsm file types may be used as database file input. If the input file is password protected, select the **Include password** check box and enter the required password in the corresponding field. To select a specific table within the input file to export data to, click the **Table** drop-down arrow and select the required table. To replace nulls in the table, select the **Replace nulls with default values (empty strings, zeros, and default dates (1/1/1970))** check box.

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		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. **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. **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. **Delimiter character** — specifies the delimiter (i.e., comma, tab, semicolon, pipe, space, other) used in the file to separate values. 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. **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. **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." **Character set** — specifies the type of encoding to use when interpreting data. Lines to skip at start — the number indicated in this box instructs Automator to skip n lines before beginning data import.

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
Report Warehouse Input	Reports from the Report Warehouse document type	❑ **Name** — name of the graphic element. ❑ **Document type** — Report Warehouse data container the report input is taken from. Specify the document type of the reports to be considered as input. You can search for documents using the following search criteria: ❑ **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. ❑ **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 *.dmod* or *.xmod* extension	❑ **Name** — name of the graphic element. ❑ **Model Path** — path that shows the File Model location. The **Model Path** parameter can be filled out manually or using **Browse**. 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: • **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). **NOTE:** Model files in compressed archive format (e.g., zip and gzip) can be used and specified. 4. Click the **OK** button.
Web Model	Model file located in the web	❑ **Name** — name of the graphic element. ❑ **Model Address** — URL of the source model file. **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.
Server Library Model	Model within a database	❑ **Name** — name of the graphic element. ❑ **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. **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.

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.	❑ **Name** — name of the graphic element. ❑ **Workspace Path** — path to the required input file location. Actions to select the workspace path: 1. Click the **Browse** button to open the Browse dialog.

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		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**. ❑ **Validate Workspace** – Click this button to validate the workspace manually. The result of workspace validation shows the following status: • Validation successful • Validation failed • Validation is not completed ❑ **Load Plans** — the list of available export load plans, including: • **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. **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. **NOTE:** DatabaseInput is required for Excel formats and must include the extension even with wildcards. **NOTE:** Additional options are available for Excel Worksheet Design. They are detailed here. • 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
		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. **NOTE:** Additional options for SQL Server, OLE DB, and Oracle source types are provided below. • **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.	❑ **Name** — name of the graphic element. ❑ **Load Plan to be Exported** — select a load plan for exporting. 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. ❑ **Export Destination** — path to required export file. ❑ **Export Type** — the type of export file. Further settings depend on the type of file and are described as advanced options below. **NOTE:** You may also use the naming macros to name the exported files and destination folder. To

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		do this, click the **Insert Macro** icon ( ) and select the naming macros from the drop-down list. **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.

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

- Greek (ISO)
- Turkish (ISO)
- Estonian (ISO)
- Latin 9 (ISO)
- Japanese (JIS)
- 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	❑ **Name** — name of the graphic element. ❑ File inputs processing: • **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
		processed by all elements after the Hub separately **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. The same change can be seen for the export elements when the **Individually (Export Only)** option is selected. For example, becomes . 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. 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). ❑ Order in which to process multiple files: • **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)	❑ **Name** — name of the graphic **element**. ❑ Path — path to the required export file location. **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. Actions to select export file path:

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		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, Web,** or **Named Paths**). 3. Depending on the selected data source type, specify the other parameters: • **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). 4. Click the **OK** button. **NOTE:** You may also use 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. ❑ **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** — 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. If the export file has PRF or PDF file type, the ADVANCED OPTIONS appear. Detailed information on the **Advanced options** (both for PRF and PDF) is specified below this table.
Table Export	Exports table data	❑ **Name** — name of the graphic element. ❑ **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
		specify the Automatic Naming options: • **By files** — produces a file for each filter (set as default) • **By tables** — produces a table for each filter ❑ **Sort** — sorting order to the data you are exporting, it is selected from the drop-down list. ❑ **Output type** — defines how the output will be produced, includes two options: • File system • Database **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. Detailed information on the **Connection options** and **File options** is specified below this table.
Summary Export	Exports summary data	**NOTE:** Settings of the Summary Report element are disable if the model is not specified or does not have any summary. ❑ **Name** — name of the graphic element. ❑ **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: • **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. ❑ **Drill State** — you can select the following drill states: • **Most detailed** — exports the summary in the most detailed (i.e., drilled down) drill state. • **Current** — exports the summary in the current drill state. ❑ **Measure** — is used to export summary that contains multiple measures. You can select the following options: • **Active measure or measure grouping** — exports only the current

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		measure or measure grouping. • **All measures as separate exports** — exports 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. ❑ **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: • **By files** — produces a file for each filter (set as default) • **By tables** — produces a table for each filter ❑ **Sort** — sorting order to the data you are exporting, it is selected from the drop-down list. ❑ **Output type** — defines how the output will be produced. You can select the following options: • File system • Database **NOTE:** If you select **File system**, you must specify **File options**. If you select **Database**, you must specify **Connection options**. Detailed information on the **Connection options** and **File options** is specified below this table.

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	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. 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. If they have not, the operations defined by red connectors will run.	• **Name** — name of the graphic element. • **File Path** — path that shows the File Prerequisite location. The **File Path** parameter can be filled out manually or using **Browse**. Actions: 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: • **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	• **Name** — name of the graphic

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
	runs the specified command line and tests that it returns the given success code. When evaluating this prerequisite, the server runs the command in a hidden window and checks that its return code matches the expected success code. By default the success code is 0 (zero), which is the default success code for programs if they complete successfully.	element. • **File Path** — path that shows the Command Line Prerequisite location. The **File Path** parameter can be filled out manually or using **Browse**. Actions: 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: • **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.	❑ **Name** — name of the graphic element. ❑ **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. ❑ **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. ❑ **Check file consistency** — specifies that

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		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. ❑ 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	Automator supports verification of report inputs on a process-by-process basis. **NOTE:** Verification is not available for database inputs. If a process 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 while processing. When verification is enabled, Automator may verify field boundaries, data types and filter results.	❑ **Name** — name of the graphic element. ❑ **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. ❑ **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. ❑ **Unfiltered table has at least one row** — checks that the unfiltered table contains at least one row of data. ❑ **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. **NOTE:** This feature is not available for database and PDF inputs. ❑ **Text field lengths** — checks that all text fields have valid field length. ❑ **Numeric field decimals** — checks that all numeric fields have valid decimals. ❑ **Numeric field lengths** — checks that all numeric fields have valid field length. ❑ **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

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		that all date/time fields have the correct format precision. When you select a verification option, verification is activated. **If verification fails setting** ❑ **Continue**. Automator will continue processing the associated actions for all inputs even if a verification failure of the specified kind is encountered. ❑ **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. ❑ **Stop**. The first failed verification causes the Verify Model to immediately fail. ❑ **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." **NOTE:** To use input files that were not verified successfully, the Verify Model element should be connected to another element using the green line.
Verify Workspace	Automator allows results in a workspace to be verified so that users can stop, proceed, or proceed with notification.	❑ Name — name of the graphic element. ❑ Load Plans to be verified — select the load plan/s for verification. The user can select several load plans by ticking their corresponding checkboxes. To export all load plans, select the **All Load Plans** checkbox. **Verification options** — choose the verification to be done: ❑ Verify DataSource availability – select this option to check whether the data source specified is available. ❑ Check that number of rows is – select this option to check if the number of rows meets the specified value: • 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 **If verification fails setting**

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		❑ **Continue**. Automator will continue processing the associated actions for the workspace even if a verification failure of the specified kind is encountered. ❑ **Continue with verified only**. Automator will continue processing the associated actions for the workspace only if it was verified successfully. ❑ **Stop**. The first failed verification causes the Verify Workspace to immediately fail. ❑ **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	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	❑ **Name** — name of the graphic element. Acceptable distribution actions: ❑ **Log** — allows you to create an event written to WindowsApplication Event Log. ❑ **Email** — allows you to email files to another user ❑ **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 ❑ **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 ❑ **Delete** — allows you to delete all files specified by the distribution ❑ **RSS** — allows you to "pull" information as opposed to the "push" model of Email ❑ **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. ❑ **Export to Amazon S3** – allows

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		you to export input and export files to Amazon S3 Detailed information on distribution settings is specified in the **Distribution Settings** table below.
Script	Automator supports the use of Visual Basic.NET script. You can find collection of Visual Process Designer scripts is the Process Designer Scripting API section.	❑ **Name** — name of the graphic element. ❑ **Script field** — displays the current script. ❑ **Run** – runs the current script ❑ **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. ❑ **User Scripting Help** — opens Help on user scripting. If the scripts have errors, all the errors are displayed below the Script field. All namespaces and references, which specified in the Global Scripts of Standard Processes, are imported in scripts of Visual Processes. 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. You can use Visual Studio.NET to help create your scripts, or you can obtain the free Microsoft .NET Framework SDK. 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. **NOTE:** Automator users can only edit scripts if they are granted editing privileges by the Admin.
Credentials	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.	❑ **Name** — name of the graphic element. ❑ **HTTP Credentials** — specify a name in the User name field and password in the Password field. **NOTE:** To use the current windows login, select the corresponding check

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		box. ❑ **FTP/SFTP Credentials** — specify a name in the User name field and password in the Password field. ❑ **Amazon S3 Credentials** — specify an ID in the Access key ID field and an access key in the Secret access key field.
Timer	When configuring a process, one of the settings you can specify is a process retries setting. 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. 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. **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.	❑ **Name** — name of the graphic element. ❑ **Retry Type**: select one of the following options: • **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. ❑ **Retry every** — the desired retry interval in hours, minutes or seconds.
Visual Process	This functionality allows processes to be run in a chain. **Example:** ❑ Add two processes. ❑ Connect the two processes. ❑ Run the new process. 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.	❑ **Name** — name of the visual process ❑ **Process name field** — click the Browse button to select a saved visual process ❑ **Description** — description of selected visual process ❑ **Preview Process** — click this option to preview the selected visual process
Standard process	This functionality allows processes to be run in a chain. **Example:** ❑ Add two processes. ❑ Connect the two processes. ❑ Run the new process. 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	❑ **Name** — name of the visual process ❑ **Process name field** — click the Browse button to select a standard process ❑ **Description** — description of the selected standard process ❑ **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.	❑ **Input** – select or multiple file inputs or web inputs to be bursted **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). ❑ **Recognize Line Key**, **Offset** – optional identifier within the input files to be bursted; bursting proceeds when the key is recognized within the input file; *offset* specifies the number of characters to be counted from the left of the key ❑ **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) ❑ **Burst Type** (Defined or Pages) **Defined**: Used when the input file contains a table. The table is parsed according to the following definitions: • **Index Table Start Maker, Offset** – an indicator that marks where in the input file the index table begins • **Line Validator, Offset** – specifies the symbols that mark the beginning of a line • **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 • **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 • **Index Table End Marker, Offset** – an indicator that

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		marks where in the input file the index table ends • **Content Start Marker, Offset** – an indicator that marks where in the input file the content of the table begins **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. • **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 ❑ **Destination Path** – location of generated files after bursting ❑ **Output Filename Mask** – name pattern for the generated files • **Header Output Mask** – this option is available in Defined Bursting; specifies name patter for the generated file containing table header information ❑ **Error Directory** – location of generated files containing errors after bursting **NOTE:** All burst output files will inherit the "last modified" timestamp of the original parent input report.

Distribution Settings

The table below specifies distribution actions.

DISTRIBUTION ACTION	DESCRIPTION	PROCEDURE AND COMMENTS
Log	Allows you to create an event written to	❑ **Type**: Select the type from the

DISTRIBUTION ACTION	DESCRIPTION	PROCEDURE AND COMMENTS
	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.	drop-down list. ❑ **Event ID**: Enter the event ID. ❑ **Message**: Type the message in the text field. **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. **NOTE:** When exporting to Knowledge Studio, all three files (*.kdd, *.ksd, and *.ksd-data) are listed individually in the event log.
Email	Allows you to email files to another user	Emailing procedure: 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. **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. 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: • 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. **NOTE:** Please specify the **Admin Host** and **Port** information in the System Settings. 8. Select the **One file per mail** check

DISTRIBUTION ACTION	DESCRIPTION	PROCEDURE AND COMMENTS
		box to attach only one file per mail. **NOTE:** When exporting to Knowledge Studio, all three files (*.kdd, *.ksd, and *.ksd-data) are kept together and sent in one email. 9. Select the email **Priority** from the following: **Low**, **Normal**, **High**.
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.	Moving procedure: 1. 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. • 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. 2. In the **Rename 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. 3. Select the **Change extension to** check box to change file's extension. Specify new extension in the enabled text box below. 4. Select the **Zip** check box, if you want to archive the file. The **Zip Options** settings appear. **NOTE:** The **Zip** check box is disabled if **Change extension to** check box selected. Under **Zip Options**, provide the following information: • 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
		store directory path. • Select the **Set password** check box to set the password. 5. In the When destination files exist option: • 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. **NOTE:** When exporting to Knowledge Studio, all three files (*.kdd, *.ksd, and *.ksd-data) are moved and listed in the log individually.
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.	Copying procedure is similar to moving procedure (see above). **NOTE:** When exporting to Knowledge Studio, all three files (*.kdd, *.ksd, and *.ksd-data) are copied and listed in the log individually.
Delete	Allows you to delete all files specified by the distribution.	**NOTE:** When exporting to Knowledge Studio, all three files (*.kdd, *.ksd, and *.ksd-data) are deleted and listed in the log individually.
RSS	Allows you to "pull" information as opposed to the "push" model of Email. RSS (Really Simple Syndication) is a widely used format for publishing information. 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.	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. • 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. 2. Select the **Create if not found** check box if you want to create the file if the specified file is not found. 3. In the **Title** box, enter the item title. 4. 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

DISTRIBUTION ACTION	DESCRIPTION	PROCEDURE AND COMMENTS
		the **Insert Macro** icon and select the naming macros from the drop-down list. 5. In the **Link** box, enter the name of link to the file. 6. Expand the **Channel** section to provide the channel parameters: • **Title**: Enter channel title. • **Channel Description**: Provide channel description. • **Link**: Enter the name of the 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. **NOTE:** When exporting to Knowledge Studio, all three files (*.kdd, *.ksd, and *.ksd-data) are listed in the RSS file individually.
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.	1. **From:** Type the address or click the **Contacts** icon to choose a sender. 2. **To:** Type the address or click the **Contacts** icon to choose a sender. **NOTE:** When typing multiple addresses, separate them with semicolon. 3. **Recipients:** Allows you to set the recipients of the e-mail, based on key. 4. **Subject:** The subject line for the message. You can use the adjacent arrow icon to insert a naming macro. 5. Type the message in the **Message** box. 6. Select the **Attach files** check box to attach a file for the distribution. 7. Select the **Zip** check box to archive the attached files. The **Zip Options** appear.

DISTRIBUTION ACTION	DESCRIPTION	PROCEDURE AND COMMENTS
		8. Under **Zip Options** provide the following information: • 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	Allows you to export input and export files to Amazon S3 **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).	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: • 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. **NOTE:** The Zip check box is disabled if Change extension to check box is selected. Under **Zip Options**, provide the following information: • 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	❑ **Name** — name of the graphic element. ❑ **Connection String** — the string for connecting to the Hive database. ❑ **Table name** — name of the Hive table. ❑ **Server** — server where the table is stored. ❑ **Directory** — directory where the table is stored. ❑ **User Name** — name of the creator of the Hive table. **Limitation**: For this output type, you must use export data in the CSV format.
Report Warehouse Output	For uploading data into a Report Warehouse document type	❑ **Name** — name of the graphic element. ❑ **Document type** — Report Warehouse data container the report input is uploaded into. ❑ **Document Group** — name of the group of document types. ❑ Processing Type: • **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). **ADVANCED OPTIONS** ❑ **Store Original Report**: When reports are

ELEMENT NAME	DESCRIPTION	PROPERTIES AND COMMENTS
		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. ❑ **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. ❑ **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. Redaction settings are available only if a redaction license is enabled in Monarch Server. ❑ **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. ❑ **Store mining data redacted**: Select this option to allow pre-mined data to be stored with redaction. **Limitation:** For this output type, you can use data from text input or report export.

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: *IList<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 it's 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 ± Export Log to export the log to an XML file.

Click the **Back to list** button Back to list to navigate to the Logs list.

Click the **Go to Process** button Go to Process 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 Previous Next 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 ✖ 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 &[inputname]s are replaced by empty strings.
Output File Location	&[output]	Full path and filename of an export file. To specify multiple output files use the convention &[output.1] &[output.2] etc. If there are more &[output]s than export files, the extra ones are replaced by empty strings.
Distribution Source file path	&[source]	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)	&[source.option]	The option specifies which part of the path to return, i.e., dir for directory, file for filename with extension, name for filename alone, ext for extension.
Job Number	&[jobid]	Job number
Job Log	&[joblog]	Full path and filename of the Job Log
Process Name	&[process]	Process name
Date Format	&[date.<date increment>.<output format>]	Subtracted date; **<date increment>** could be -1d, +1d, -1m, +1m, -1y, +1y, and **<output format>** could be yyyyMMdd, yyyy-MM-dd, etc. The output format is case-sensitive. For a full list of supported format specifiers, click here.
System-Wide Counter	&[counter]	System counter. Automatically increments with each use — counts across all jobs in all processes.
Item Addressing (Advanced)	&[item.xpath]	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	&[source]	Full path of an item.
Item File Directory	&[source.dir]	Full path of item directory.
Item File Name	&[source.file]	File name of an item.
Item File Name (no ext.)	&[source.name]	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] &[item.description] &[item.sort] &[item.filter] &[item.summary] &[item.totals/@record_count	Returns the auto-generated item title Returns the auto generated item description Returns the sort, if specified Returns the filter, if specified Returns the summary, if one exists 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
&[dd.+1d]	The day portion of tomorrow's date.
&[dd.+d]	Synonymous with &[dd.+1d]. The count defaults to 1.
&[mm.+1d]	The month portion of tomorrow's date.
&[yyyy.+1d]	The 4-digit year portion of tomorrow's date.
&[dd.-1d]	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>
    <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

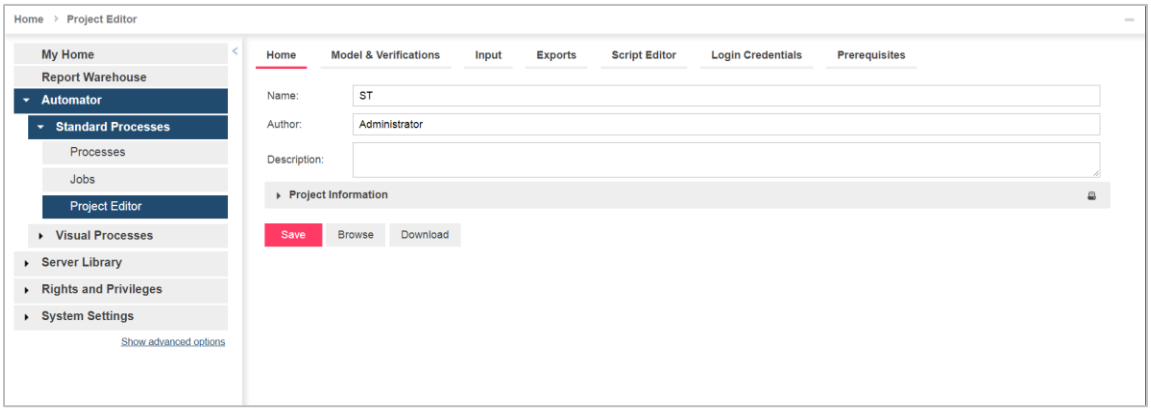
The process name in the XML file (“m” in the example above) should be the same as the name of the Standard Process created.

Other than XML, the following JSON structure may also be used:

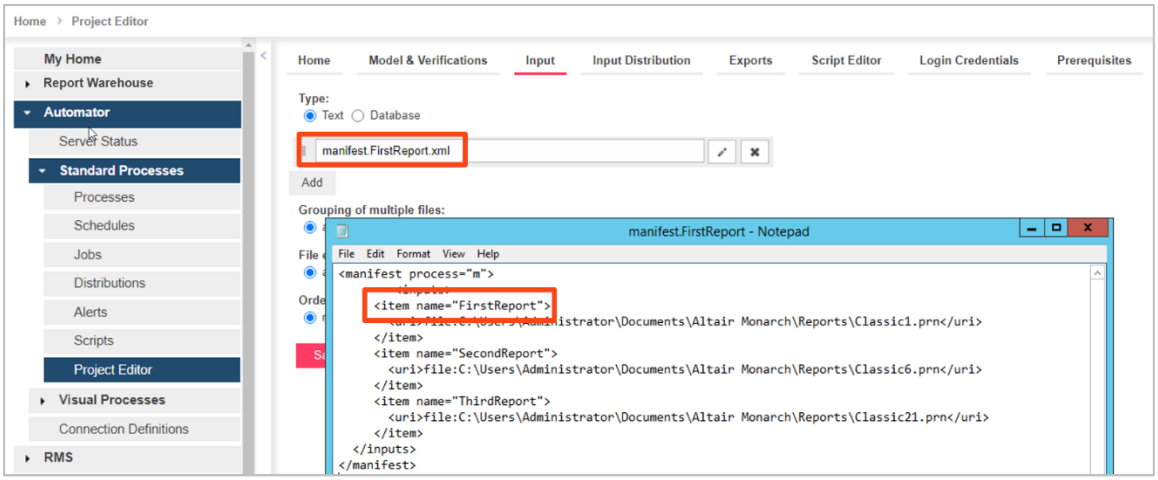
```
{
"process": "ProcessName",
"inputs": [
{
"name": "InputName",
"uri": "Input path"
}
]
}
```

To proceed with manifest monitoring

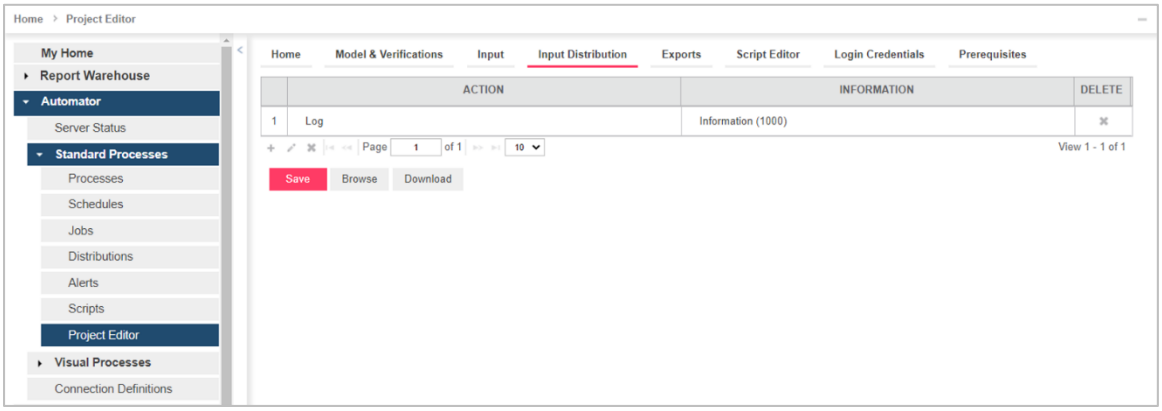
1. Open **Project Editor**. Create a new project, and assign it with any name.



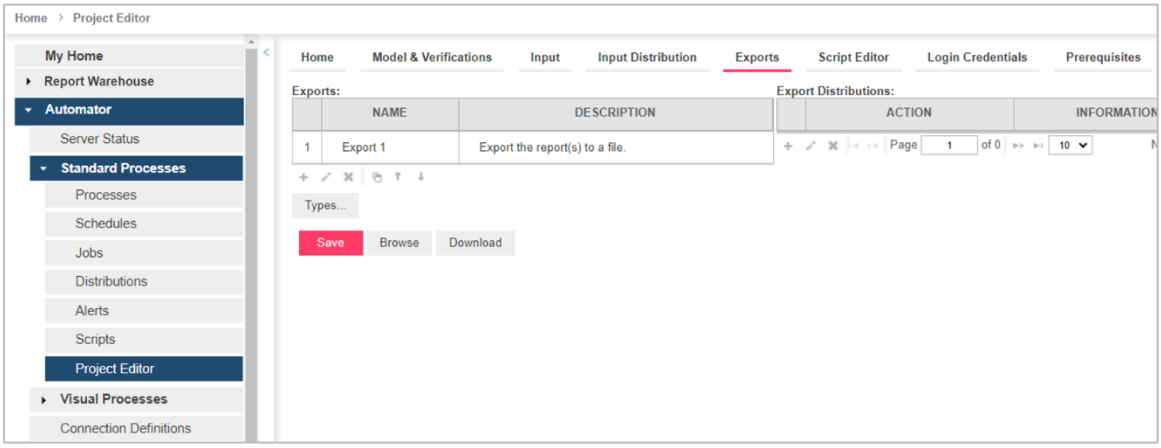
2. Open the **Model & Verifications** tab, and set the model for the new project. See [Specifying Model Settings for a Project File](#).
3. Open the **Input** tab to add an input file. If you would like to use an xml file as the input file, rename the file as “manifest.xxx,” where “xxx” is the item name from the xml file.



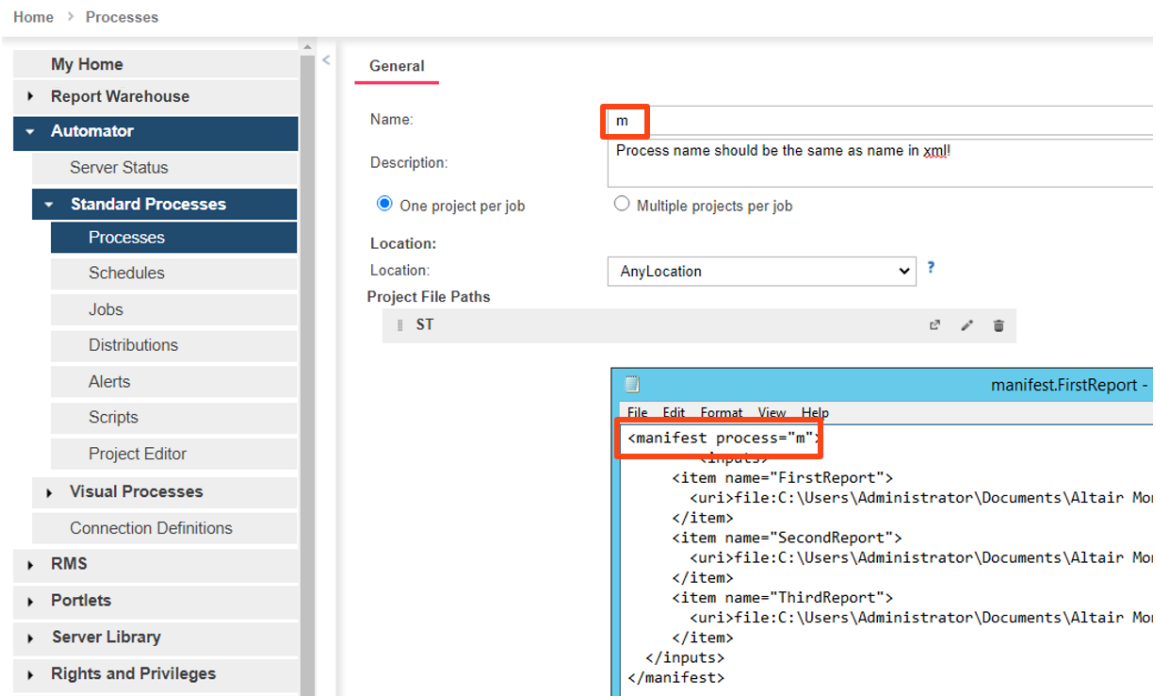
4. Add any Input Distribution.



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**.

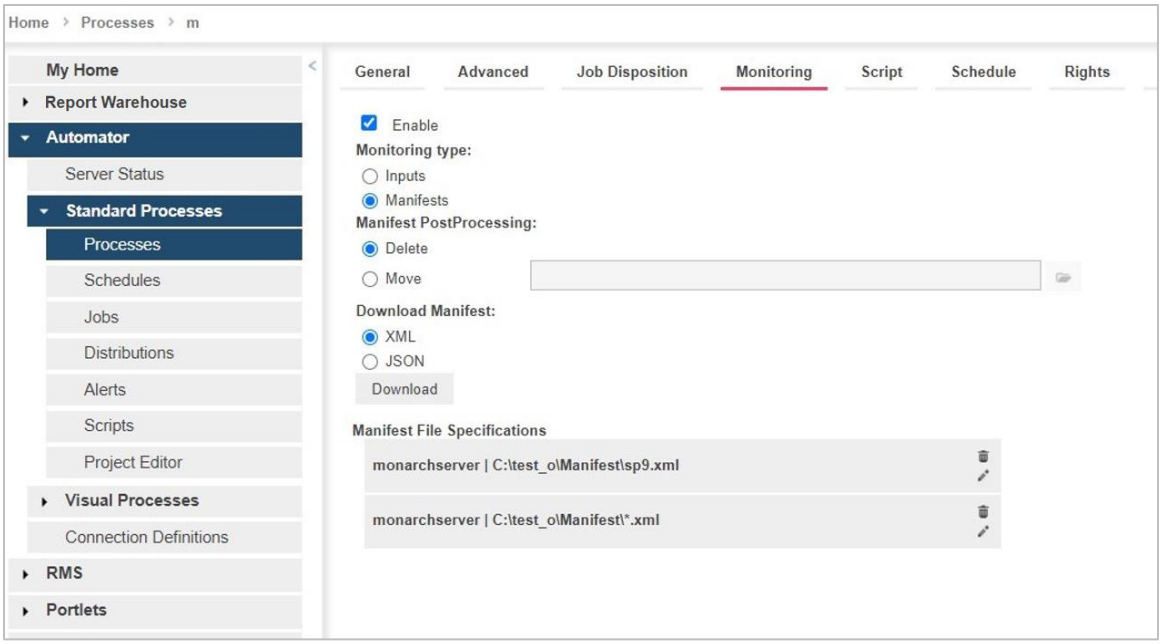
> **NOTE** The process name in the XML file ("m" in the example above) should be the same as the name of the Standard Process created.

8. Open the process you created in Step 6.

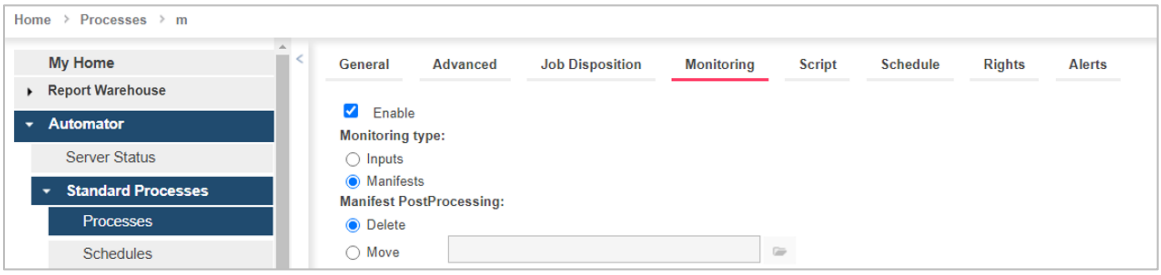
   In the **Monitoring** tab, click the **Enable** check box, and select **Manifests** under **Monitoring Type**.

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

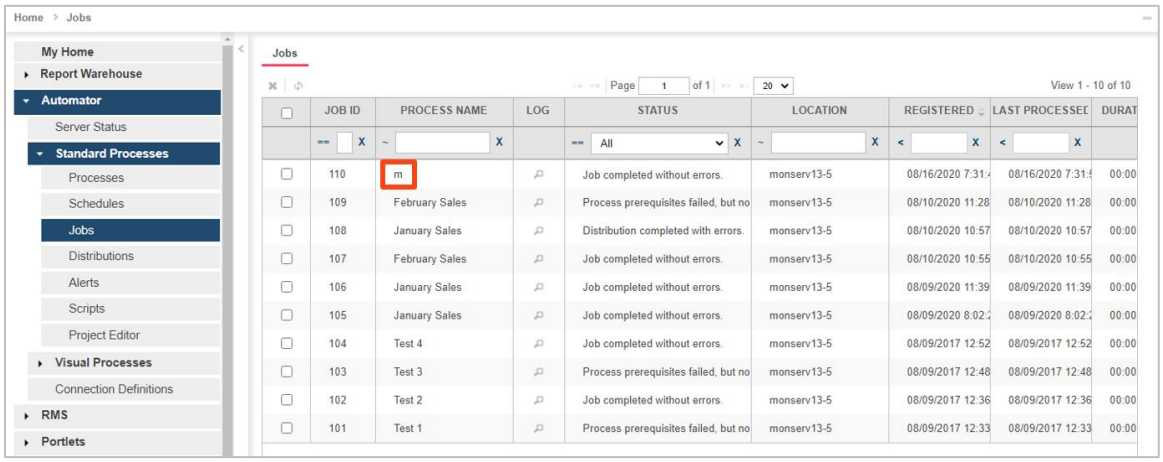


11. **Save** the process.
12. Proceed to the **Jobs** page under **Standard Processes** to view the job log or re-run the job.

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.



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

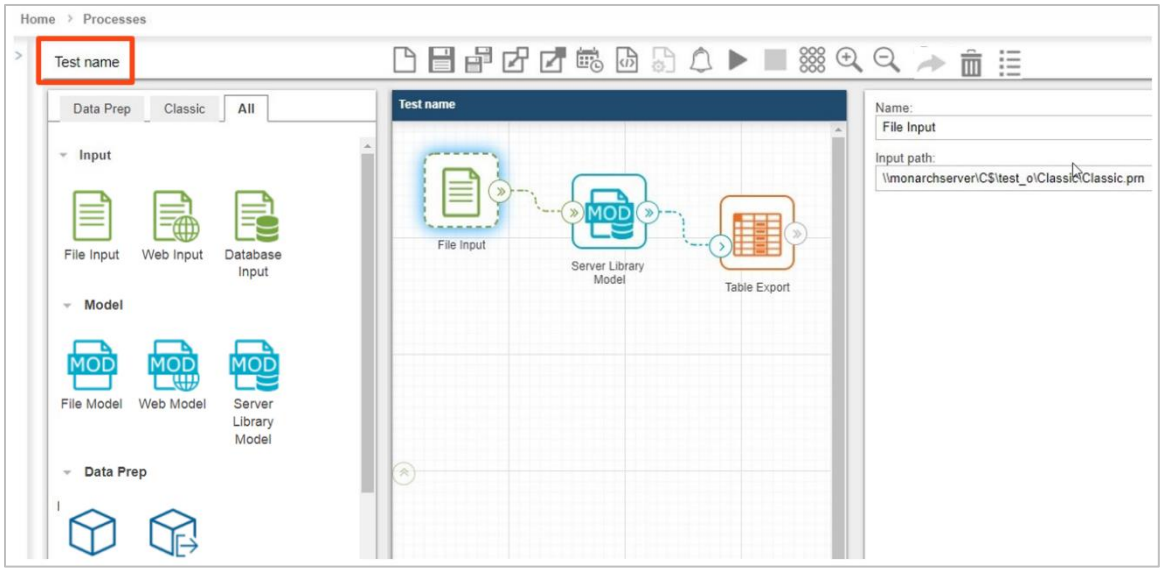
- The process name in the XML file (“Test name” in the example above) should be the same as the name of the Visual Process created.
- The following JSON structure may also be used:

```
{
"ProcessName": "Test name",

"ProcessItems": [
    {
      "ItemType": "InputFile",
      "FileInput": {
        "FileLocation": "Input path",
        "Name": "File Input",
  "PdfPassword": "password"
      }
    }
]
}
```

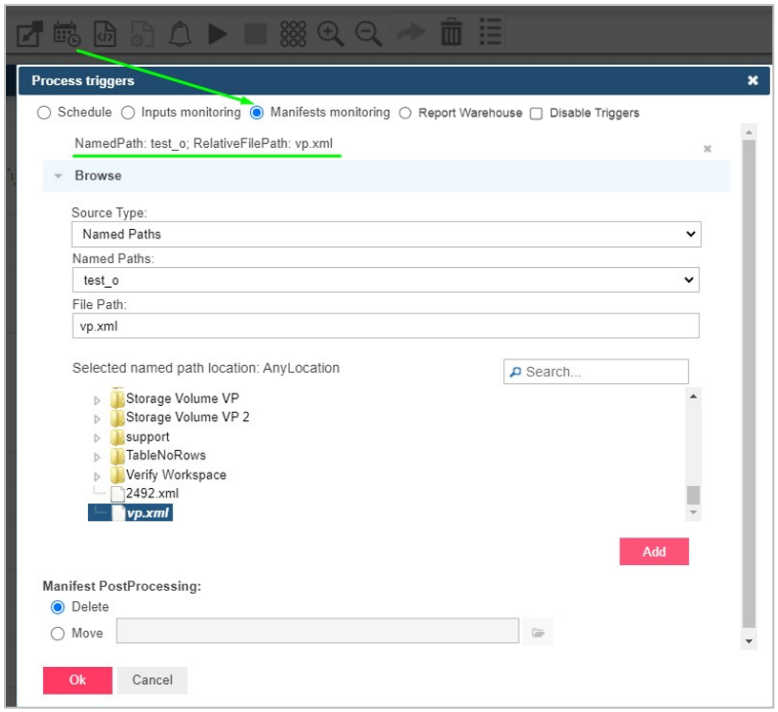
To proceed with manifest monitoring

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.prn,” and the XML file could have the input file “Classic1.prn.”

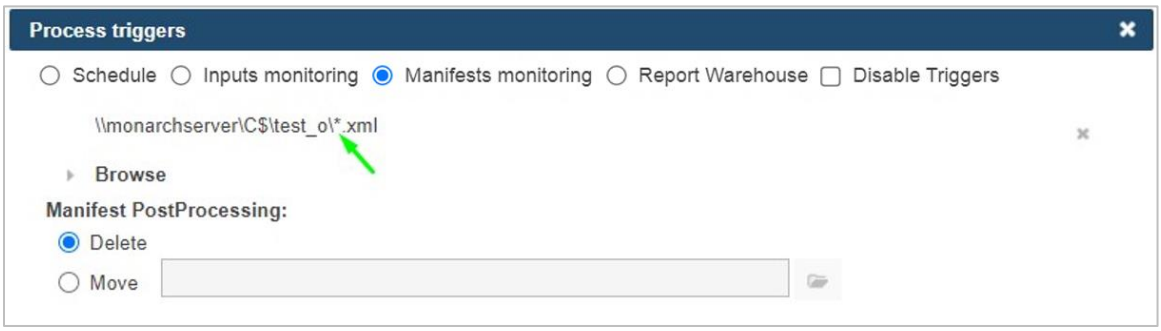
2. Click the **Triggers** icon to launch the **Process triggers** window.
3. Select **Manifests monitoring** (for details, see Manifests Monitoring Trigger).
   a. Click **Browse**.
   b. Select the **Source Type**, and add the manifest xml file you prepared (as described in the introduction of this section).
   c. 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).

d. 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

e. Click the **Ok** button to save the process monitoring.

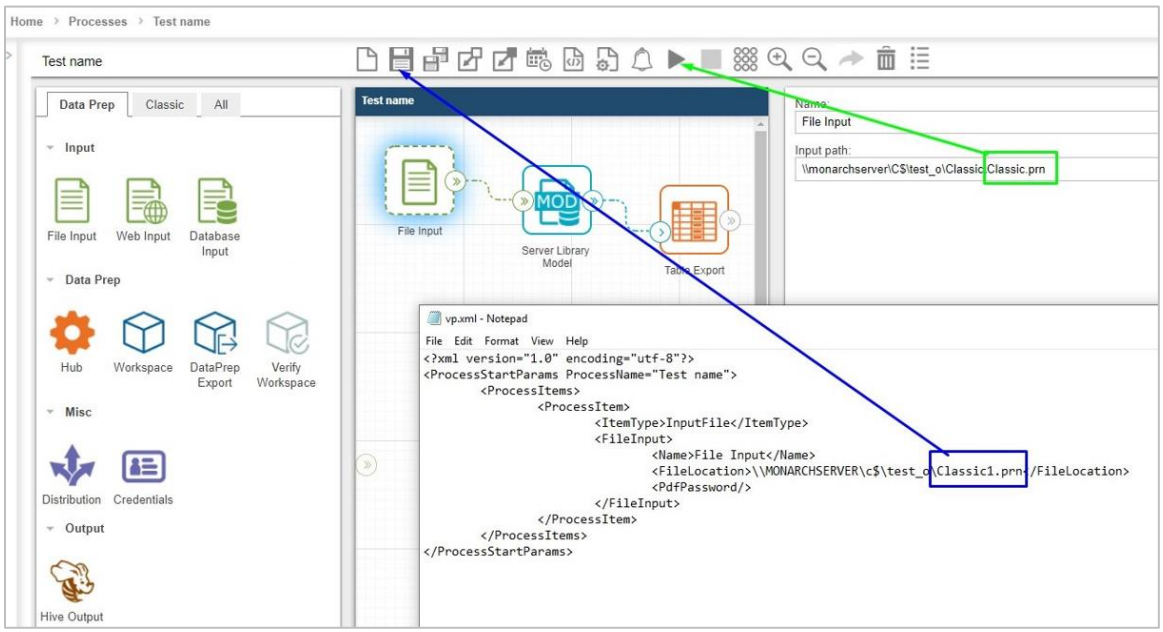


4. Save the process.

NOTE If you are using the input file specified in the xml file you created in the beginning

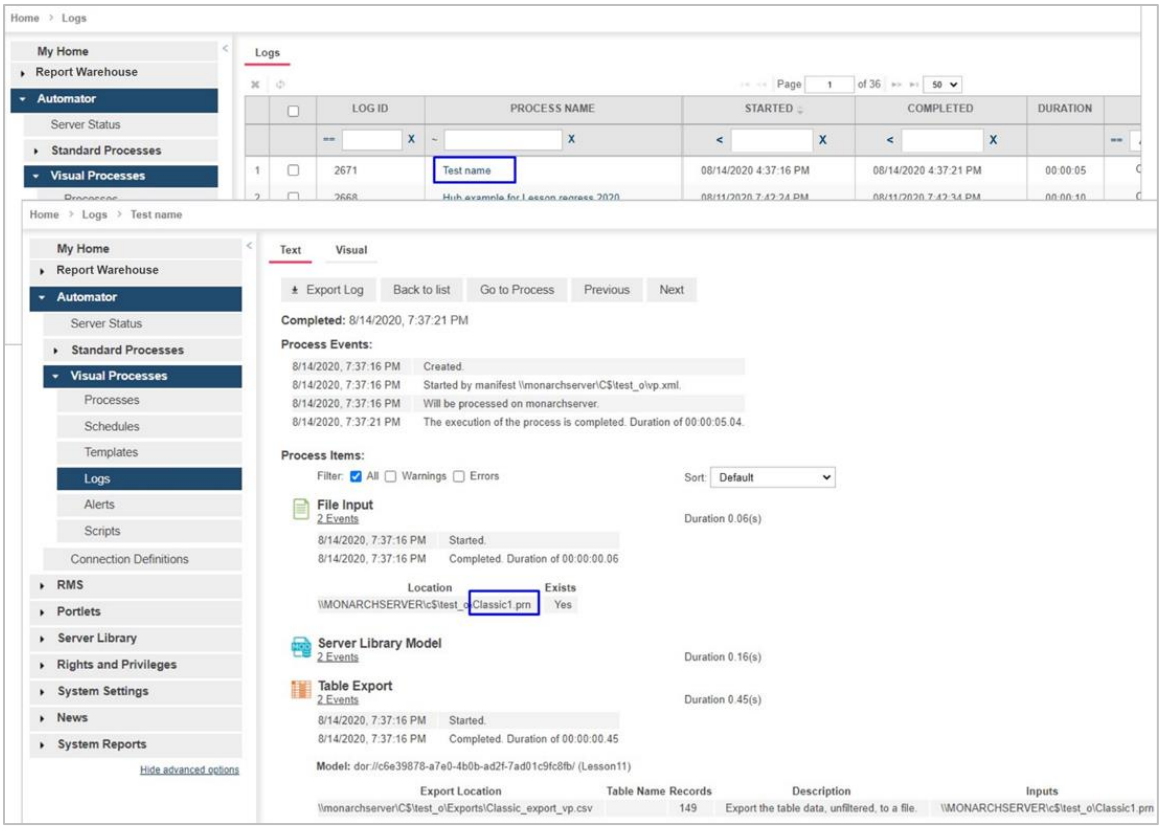
(*Classic1.prn* in the manifest xml example above), click the **Save** icon only. When the xml file is found, the process will run automatically.

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.



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:sequence>
      <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:sequence>
      <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: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: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: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>
```

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 MSClient 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 MSClient 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 MSClient 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

FOLDERS

You can create server library folders that include particular project, models and templates.

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** filed.
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.
  - **SubAdmin**: Allows users limited administration.
  - **Portlets**: Allows users to view portlets — Panopticon Workbooks — in Client.
  - **SaaS Administrator**: Allows users limited administration.
- Report Warehouse privileges
  - **Document search**: Allows users to configure searches.
  - **Save search**: Allows users to save searches that they created.
  - **Report search**: Allows users to search and view original reports.
  - **Delete document**: Allows users to delete documents from the report warehouse.
  - **Download Original Report**: Allows users to download the archive copy of reports.
  - **Concatenate Original Reports**: Allows users to download a number of original PDF reports as a single file.
  - **Create annotation**: Allows users to create annotations.
  - **Create notification**: Allows users to create search notifications and filing notifications.
  - **See trustee list on save Annotation**: Allows the user to view the list of trustees in the saved annotation report.
  - **See trustee list on save Search**: Allows the user to view the list of trustees in the saved search.

- **Add attachments on Annotations**: Allows the user to add attachments to the annotation report.
- **Create Annotations for all users**: Allows the user to create an annotation report accessible to all users.
- **Edit and delete Annotations**: Allows the user to modify and remove annotations.
- **See Annotations**: Allows the user to view the existing annotations.

- 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.

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.

SubAdmin

Only the following components are available to the **SubAdmin** user:

- Home
- Maintenance Tasks: Delete, Purge, Re-Index, Pre-Mined Data, Block
- Maintenance Processes
- Index Fields
- Document Groups
- Document Types (only rights and edit actions are available, and Copy Profile)
- All Document Retrieval options are available except Offline Processes.
- Portlets
- Server Library
- All System Reports except Removed Reports are available.
- All News are available except Global News
- Help

SaaS Administrator

SaaS admin: administrator at the remote customer site without knowledge of paths and physical system structure.

The **SaaSAdmin** user cannot:

- Change configured physical attributes in MES.
- Create or manage Storage Volumes or Storage Groups.
- Define or view post-processing delete/moves/errors.
- Establish Physical and Location resources in System Preferences/System Defaults.
- Create admins, sub admins, or group admins.
- Have visibility to Trustees of type (Full) Admin.
- Use the full SQL Syntax (only SELECTs are available.)

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:
   - **Report Security**: Allows to grant access to report security rules for the role, user group or user.
   - **Document Types**: Allows to grant access to document types for the role, user group or user.
   - **Unredacted Document Types**: Allows you to select document group and assign its document types for the role, user group or user.

> **NOTE** The **Unredacted Document Types** tab becomes available only if a redaction license is enabled in MS.

   - **Searches**: Allows to grant access to searches for the role, user group or user.
   - **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.
   - **Search Notifications**: Allows to grant access to search notifications for the role, user group or user.
   - **System Notifications**: Allows to grant access to system notifications for the role, user group or user.

- **Filing Notifications**: Allows to grant access to filing notifications for the role, user group or user.
- **Group Administrator**: Allows to assign limited administration rights to group administrators.
- **Security Keywords**: Specifies which security keywords apply to the role, user group or user.
- **Global Security Keywords**: Specifies which global security keywords apply to the role, user group or user.
- **Document States Security**: Allows to assign rights of managing document states to 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.

Group Administrator

An administrator can assign limited administration rights to group administrators (users who can perform a limited set of operations in Web Admin). Limited administration rights can be assigned to a user, group, or role. In case of a group or role, all explicit and implicit members of that group or role will have the specified limited rights.

Group administrators can view only those objects, to which they have corresponding rights. For example, when defining a new filing task, they can work only with document types assigned to them.

To display the **Group Administrator** page, when assigning access rights, click the **Group Administrator** tab at the top of the page.

The **Group Administrator** page contains the following items:

- **Privileges**: Select check boxes to assign the following privileges to a group administrator:
  - **Manage Filing Task**: This privilege enables a group administrator to:
    - Create a new filing task. When creating a filing task, group administrators can view only those document types, for which they have rights.
    - Edit existing filing tasks for document types, for which a group administrator has rights.
    - Delete existing filing tasks for document types, for which a group administrator has rights.

> **NOTE** The group administrator can not add a new location.

  - **Manage Searches**: This privilege enables a group administrator to:
    - Create a new search using document types, for which a group administrator has rights.
    - Edit existing searches, for which a group administrator has rights. The Search Definition page shall contain only document types, for which a group administrator has rights.
    - Delete existing searches, for which a group administrator has rights.
    - Assign rights to any user, group, or role for existing searches, for which a group administrator has rights.

- **Manage Maintenance Process**: This privilege enables a group administrator to:
  - Create a new maintenance process.
  - Edit any existing maintenance processes (including schedule type, periodicity, and suspension flag).
  - Delete any existing maintenance processes.
  - Include to or remove from a maintenance process any maintenance tasks.
- **Manage News**: This privilege enables a group administrator to:
  - Create global and group specific news.
  - Edit global and group specific news.
  - Delete global and group specific news.
  - Delete expired news.
  - Assign rights to any user, group, or role for existing group specific news.
- **Manage Maintenance Task**: This privilege enables a group administrator to:
  - Create a new maintenance task. When creating a maintenance task, group administrators can view only those document types, for which they have rights.
  - Edit existing maintenance tasks for document types, for which a group administrator has rights.
  - Delete existing filing tasks for document types, for which a group administrator has rights.
- **Manage Notifications**: This privilege enables a group administrator to:
  - Create a new notification using searches, for which a group administrator has rights.
  - Edit existing notifications, for which a group administrator has rights. The Notification Definition page shall contain only searches, for which a group administrator has rights.
  - Delete existing notifications, for which a group administrator has rights.
  - Assign rights to any user, group, or role for existing notifications, for which a group administrator has rights.
- **Manage Document Groups**: This privilege enables a group administrator to:
  - Create a new document group.
  - Edit existing document groups.
  - Delete existing document groups.
  - Include or remove from existing document groups any document types, for which a group administrator has rights.
- **Manage Document Types**: This privilege enables a group administrator to:
  - Create a new index field.
  - Edit existing index fields.
  - Delete existing index fields.
  - Upload a new model.
  - Edit existing models.
  - Delete existing models.
  - Create a new document type.
  - Edit existing document types the group administrator has rights for.
  - Delete existing document types the group administrator has rights for.
  - Assign rights to any user, group, or role for existing document types the group administrator has rights for.

        - Manage templates of existing document types the group administrator has rights for:
            - Upload a new template.
            - Edit existing templates the group administrator has rights for.
            - Delete existing templates the group administrator has rights for.
            - Assign rights to any user, group, or role for existing templates the group administrator has rights for.
    - **Manage Filing Process**: This privilege enables a group administrator to:
        - Create a new filing process.
        - Edit any existing filing processes (including schedule type, periodicity, and suspension flag).
        - Delete any existing filing processes.
        - Include or remove from a filing process any filing tasks.
    - **Manage Search Groups**: This privilege enables a group administrator to:
        - Create a new search group.
        - Edit existing search groups.
        - Delete existing search groups.
        - Include or remove from existing search groups any searches, for which the group administrator has rights.
- **Manage Filters**: After you select this check box, the **Manage Filters** button becomes visible. Click it to expand a hidden section, allowing you to assign roles and groups, which a group administrator can manage. To assign a role or group, drag it from the left pane to the right pane. This privilege enables a group administrator to:
    - Manage filters of the following trustees:
        - Assigned roles and groups.
        - Any implicit or explicit members of assigned roles and groups.
    - Assign any filters based on their filters. If they are restricted from seeing a particular keyword, then any filters in the set shall also include that restriction.
- **Manage Users**: After you select this check box, the **Manage Users** button becomes visible. Click it to expand a hidden section, allowing you to assign roles and groups, which a group administrator can manage. To assign a role or group, drag it from the left pane to the right pane. This privilege enables a group administrator to manage the assigned roles and groups.
- **Save**: Saves the settings.
- **Clear**: Clears all the boxes.
- **Cancel**: Cancels the operation.
- **Copy Profile**: Opens a dialog box, where you can select a role, user or group. All its settings will be copied. You can make the necessary changes.

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.

ABOUT SECURITY KEYWORDS

Security keywords provide a way of limiting the documents within a document type that a user can retrieve.

NOTE	If you don't want a user to retrieve any documents of a particular document type, you can remove that document type from the user. Secured keywords can be assigned to roles, user groups and users. When you assign a security keyword to a role, user group or user, the specified users can only retrieve documents that meet the criteria of the security keyword. No other documents of the specified document type can be retrieved by them.

For example, you have a **Homes for Sale** document type and you specify **Home Location** = **Lowell** as a security keyword (**Home Location** being an index field, and **Lowell** being an index value) for a particular user. As a result, the user can only retrieve documents of the **Homes for Sale** document type, in which **Lowell** occurs in the **Home Location** index field. Documents with **Home Location** other than **Lowell** (for example, **Leominster** or **Fitchburg**) are not available to the user.

See also Assigning Security Keywords to Roles, User Groups or Users.

GLOBAL SECURITY KEYWORDS

Global security keywords are similar to security keywords, except that global security keywords are automatically assigned to a document type if the document type uses the keyword as an index field. This capability is automatically applied to newly created document types that use the index field.

To assign a global security keyword to a role, user group or user

1. On the Navigation Tree, click **Rights and Privileges**, and select **Roles**, **User Groups** or **Users**.
2. Click the name of the role, user group or user you want to assign global security keywords to.
3. Select the **Global Security Keywords** tab at the top of the page.
4. Click the **Add condition** button. A row of fields will appear.

5. Select a Boolean operator (**AND** or **OR**). While the **AND** operator can be used to link conditions for different keyword types, the **OR** operator can only be used to link conditions of the same keyword type.
6. Select a keyword type from the list.
7. Select a relational operator (equal to (=), not equal to (<>), less than (<), greater than (>), less than or equal to (<=), greater than or equal to (>=).)
8. Enter a keyword value. To look up available keyword values, click the **Lookup** button.
9. To add another condition, click the **Add condition** button and repeat steps 5-8. To remove a condition, click the **Remove** button.
10. Click **Save** to save the global security keyword configuration, then click **Save** on the Role (User Group or User) page to assign the global security keyword to the role (user group or user).

To get a list of documents that correspond to the saved global security keywords

1. Click the **Preview** button.
2. In the dialog box that appears, choose the document group and document type from the lists.

   The system will display a list of documents that correspond to the defined global security keywords. The list contains the following information:

   - Document name.
   - Document type.
   - Document data.

ASSIGNING SECURITY KEYWORDS TO ROLES, USER GROUPS OR USERS

When you assign security keywords to a role, user group or user, they can retrieve only those documents that meet the criteria of the security keyword. No other documents of the specified document type can be retrieved by them.

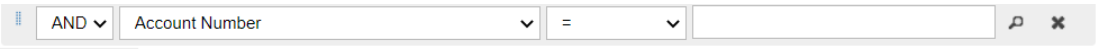
For example, you have a **Homes for Sale** document type and you specify **Home Location** = **Lowell** as a security keyword (**Home Location** being an index field, and **Lowell** being an index value) for a particular user. As a result, the user can only retrieve documents of the **Homes for Sale** document type, in which **Lowell** occurs in the **Home Location** index field. Documents with **Home Location** other than **Lowell** (for example, **Leominster** or **Fitchburg**) are not available to the user.

To assign a security keyword to a role, user group or user

1. On the Navigation Tree, click **Rights and Privileges**, and select **Roles**, **User Groups** or **Users**.
2. Click the name of the role, user group or user you want to assign security keywords to.
3. Select the **Security Keywords** tab at the top of the page.
4. From the **Document Groups** drop-down list, select the appropriate document group. The document types assigned to the selected document group will appear in the **Document Types** box.

> **NOTE** You can search for a document type by entering (part of) its name in the **Filter by** box and clicking the **Apply** button.

5. Click the **Add condition** button. A row of fields will appear.



6. Select a Boolean operator (**AND** or **OR**). While the **AND** operator can be used to link conditions for different keyword types, the **OR** operator can only be used to link conditions of the same keyword type.
7. Select a keyword type. The list contains keyword types that are used in the selected document types.
8. Select a relational operator (equal to (=), not equal to (<>), less than (<), greater than (>), less than or equal to (<=), greater than or equal to (>=).)
9. Enter a keyword value. To look up available keyword values, click the **Lookup** button.
10. To add another condition, click the **Add condition** button and repeat steps 6-9. To remove a condition, click the **Remove** button.
11. Click **Save** to save the security keyword configuration, then click **Save** on the Role Role (User Group or User) page to assign the security keyword to the role (user group or user).

To get a list of documents that correspond to the saved security keywords

1. Click the **Preview** button.
2. In the dialog box that appears, choose the document group and document type from the lists.

   The system will display a list of documents that correspond to the defined security keywords. The list contains the following information:

   - Document name.
   - Document type.

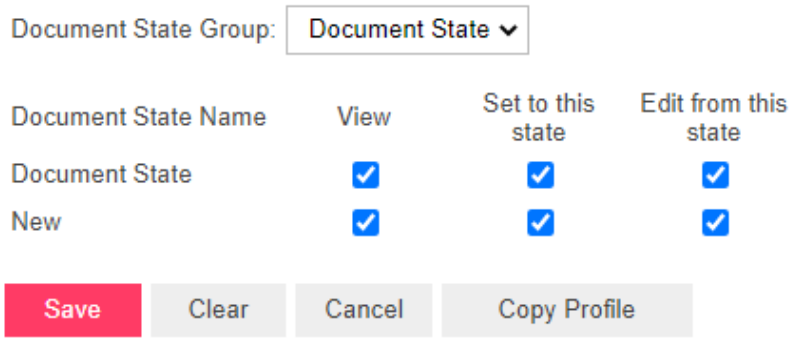
- Document data.

See also: Global Security Keywords.

DOCUMENT STATES SECURITY

To assign rights of managing document state

1. On the Navigation Tree, click **Rights and Privileges**, and select **Roles**, **User Groups** or **Users**.
2. Click the name of the role, user group or user you want to assign rights to change document state to.
3. Select the **Document States Security** tab at the top of the page.
4. From the **Document State Group** drop-down list, select the appropriate document state group. The list of document states assigned to the selected document state group appears.



5. Select the appropriate check boxes for each state:
   - **View**: allows to view the document state
   - **Set to this state**: allows to set to this document state
   - **Edit from this state**: allows to switch from this document state
6. Click **Save**.

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.
     - **SubAdmin**: Allows the user limited administration.
     - **SaaS Administrator**: Allows the user limited administration.
     - **Portlets:** Allows the user to view Panopticon Workbooks in MSClient.
     - **Document Search**: Allows the user to configure searches.
     - **Save Search**: Allows users to save searches that they created to the database.
     - **Report search**: Allows the user to search and view original reports.
     - **Delete document**: Allows the user to delete documents from the report warehouse.
     - **Download Original Report**: Allows the user to download the archive copy of reports.
     - **Concatenate Original Reports**: Allows users to download a number of original PDF reports as a single file.
     - **Create annotation**: Allows the user to annotate documents.
     - **Create notification**: Allows the user to create notifications.
     - **See trustee list on save Annotation**: Allows the user to view the list of trustees in the saved annotation report.
     - **See trustee list on save Search**: Allows the user to view the list of trustees in the saved search.
     - **Add attachments on Annotations**: Allows the user to add attachments to the annotation report.
     - **Create Annotations for all users**: Allows the user to create an annotation report accessible to all users.
     - **Edit and delete Annotations**: Allows the user to modify and remove annotations.
     - **See Annotations**: Allows the user to view the existing annotations.
     - **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.
   - **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.
   - **Group Administrator** tab allows you to grant privileges rights to group administrators. To do this, select appropriate check boxes, and click **Save**.
   - **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.

- **Security Keywords** tab allows you to assign security keywords to the current role.
- **Global Security Keywords** tab allows you to assign global security keywords to the current role.
- **Report Security** tab allows you to grant access to report security rules for the current role. To do this, select a document group and a document type, then drag appropriate items from the left pane to the right pane, and click **Save**.
- **Document Types** tab allows you to grant access to document types for the current role. To do this, select a document group from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.
- **Unredacted Document Types** tab allows you to select document group and assign its document types for the role. To do this, select a document group from the list, then drag appropriate items from the left pane to the right pane, and click **Save**. Document types from the **Assigned Document Types** pane will be assigned.

  When assigned, on the **Report** view, **Data** view, and **Summary** view data will be displayed in the unredacted format.

  For unassigned document types, data will be displayed in the redacted format.

NOTE	The **Unredacted Document Types tab** becomes available only if a redaction license is enabled in MS.

- **Portlets** tab allows you to assign portlet access rights to the current user role.
- **Searches** tab allows you to set the accessible searches for the current role. To do this, select a search group from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.
- **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**.
- **Search Notifications** tab allows you to assign search notifications 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**.
- **System Notifications** tab allows you to assign system notifications to the current role. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
- **Filing Notifications** tab allows you to assign filing notifications to the current role. 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).

- **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 ✖ 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.
  - **SubAdmin**: Allows the user limited administration.
  - **SaaS Administrator**: Allows the user limited administration.
  - **Portlets:** Allows the user to view Panopticon Workbooks in MSClient.
  - **Document Search**: Allows the user to configure searches.
  - **Save Search**: Allows users to save searches that they created to the database.
  - **Report search**: Allows the user to search and view original reports.
  - **Delete document**: Allows the user to delete documents from the report warehouse.
  - **Download Original Report**: Allows the user to download the archive copy of reports.
  - **Concatenate Original Reports**: Allows users to download a number of original PDF reports as a single file.
  - **Create annotation**: Allows the user to annotate documents.
  - **Create notification**: Allows the user to create notifications.
  - **See trustee list on save Annotation**: Allows the user to view the list of trustees in the saved annotation report.
  - **See trustee list on save Search**: Allows the user to view the list of trustees in the saved search.
  - **Add attachments on Annotations**: Allows the user to add attachments to the annotation report.
  - **Create Annotations for all users**: Allows the user to create an annotation report accessible to all users.
  - **Edit and delete Annotations**: Allows the user to modify and remove annotations.
  - **See Annotations**: Allows the user to view the existing annotations.
  - **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.
- **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.
- **Group Administrator** tab allows you to grant limited administration rights to group administrators. To do this, select appropriate check boxes, and click **Save**.
- **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.

- **Security Keywords** tab allows you to assign security keywords to the current user group.
- **Global Security Keywords** tab allows you to assign global security keywords to the current user group.
- **Report Security** tab allows you to grant access to report security rules for the current user group. To do this, select a document group and a document type, then drag appropriate items from the left pane to the right pane, and click **Save**.
- **Document Types** tab allows you to grant access to document types for the current user group. To do this, select a document group from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.
- **Unredacted Document Types** tab allows you to select document group and assign its document types for the user group. To do this, select a document group from the list, then drag appropriate items from the left pane to the right pane, and click **Save**. Document types from the **Assigned Document Types** pane will be assigned.

  When assigned, on the **Report** view, **Data** view, and **Summary** view data will be displayed in the unredacted format.

  For unassigned document types, data will be displayed in the redacted format.

> **NOTE** The **Unredacted Document Types** tab becomes available only if a redaction license is enabled in MS.

- **Portlets** tab allows you to assign portlet access rights to the current user user group..
- **Searches** tab allows you to set the accessible searches for the current user group. To do this, select a search group from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.
- **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**.
- **Search Notifications** tab allows you to assign search notifications 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**.
- **System Notifications** tab allows you to assign system notifications to the current user group. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
- **Filing Notifications** tab allows you to assign filing notifications to the current user group. 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).

- **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.
     - **SubAdmin**: Allows the user limited administration.
     - **SaaS Administrator**: Allows the user limited administration.
     - **Portlets:** Allows the user to view Panopticon Workbooks in MSClient.
     - **Document Search**: Allows the user to configure searches.
     - **Save Search**: Allows users to save searches that they created to the database.
     - **Report search**: Allows the user to search and view original reports.
     - **Delete document**: Allows the user to delete documents from the report warehouse.
     - **Download Original Report**: Allows the user to download the archive copy of reports.
     - **Concatenate Original Reports**: Allows users to download a number of original PDF reports as a single file.
     - **Create annotation**: Allows the user to annotate documents.
     - **Create notification**: Allows the user to create notifications.
     - **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.

- **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.
   - **Group Administrator** tab allows you to grant limited administration rights to group administrators. To do this, select appropriate check boxes, and click **Save**.
   - **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.

   - **Security Keywords** tab allows you to assign security keywords to the current user.
   - **Global Security Keywords** tab allows you to assign global security keywords to the current user.
   - **Report Security** tab allows you to grant access to report security rules for the current user. To do this, select a document group and a document type, then drag appropriate items from the left pane to the right pane, and click **Save**.
   - **Document Types** tab allows you to grant access to document types for the current user. To do this, select a document group from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.
   - **Unredacted Document Types** tab allows you to select document group and assign its document types for the user. To do this, select a document group from the list, then drag appropriate items from the left pane to the right pane, and click **Save**. Document types from the **Assigned Document Types** pane will be assigned.

     When assigned, on the **Report** view, **Data** view, and **Summary** view data will be displayed in the unredacted format.

     For unassigned document types, data will be displayed in the redacted format.

> **NOTE** The **Unredacted Document Types** tab becomes available only if a redaction license is enabled in MS.

- **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**.
- **Searches** tab allows you to set the accessible searches for the current user. To do this, select a search group from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.
- **Search Notifications** tab allows you to assign search notifications 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**.
- **System Notifications** tab allows you to assign system notifications to the current user. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
- **Filing Notifications** tab allows you to assign filing notifications 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 user, by clicking the **Copy Profile** button, available on the above mentioned tabs (except **Views** and **Report Security** tabs).

- **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 ✖ 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.

DOCUMENT STATE GROUPS

A document state group is a classification of document states. Each document group contains one or more document states. All document state groups used in the system are listed on the Document State Groups page.

To display the Document State Groups page, click **Rights and Privileges** on the Navigation Tree, and select **Document State Groups**.

Use this page to add, edit or remove document state groups.

Adding a Document State Group

To add a document state group

1. On the Navigation Tree, click **Rights and Privileges**, and select **Document State Groups**.
2. Click + on the **Document State Groups** page. The Document State Group page appears.
3. Enter a unique name of a new document state group in the **Name** field.
4. Enter any text defining the document state group in the **Description** field. This step is optional.
5. To add document state, click the **Add** button.
6. Enter the document state name in the **Document State** field, and then click the **Save** button. For more information, see Add new document states.
7. Click **Save**.

Editing a Document State Group

To edit a document state group

1. On the Navigation Tree, click **Rights and Privileges**, and select **Document State Groups**.
2. Select the name of the document state group you want to edit. Click the name of the document state group or click anywhere on the row, and then click . The **Document State Group** page appears.
3. Make the necessary changes
   - Edit document state group name and description
   - Add document states
   - Edit document states
   - Delete document states
4. Click **Save**.

Deleting a Document State Group

To delete a document state group

1. On the Navigation Tree, click **Rights and Privileges**, and select **Users**.
2. Click ✖ at the end of the row with the document state group you want to delete. A dialog box appears.
3. Click **Delete** to delete the document state group from the list.

Document States

Document states are used to indicate the current state of a document. An administrator defines the set of document states for each document state group. Every document state group has a fixed default state.

> **NOTE** Default state can be edited but cannot be deleted.

All document states of the particular document state group are listed on the Document State Group page. To display this page, on the Navigation Tree, click **Rights and Privileges**, then click **Document State Groups** and select necessary document state group from the list.

Use this page to add, edit or remove document states.

Adding a Document State

To add a document state

1. Click the **Add** button on the **Document State Group** page.
2. Enter the document state name in the **Add Document State** dialog box.
3. Click **Save**.

Editing a Document State

To edit a document state

1. On the **Document State Group** page, click by the name of the document state you want to edit.
2. Enter the document state name in the **Edit Document State** dialog box.
3. Click **Save**.

Deleting a Document State

To delete a document state

1. On the **Document State Group** page, click at the end of the row with the document state you want to delete.
2. Click **Delete** to delete the document state from the list.
3. Click **Save**, to save the changes.

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
- Locale settings
- Automator settings
- Server locations 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.
- **System Defaults**: Set default parameters to be used when creating new volumes and filing tasks.
- **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.
- **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.
- **Allow empty date range in documents/reports search**: Select this check box to permit empty date range in documents/reports search results list.
- **Dynamic search on lists**: Select this check box to enable the auto-suggest feature in MSClient, 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.
- **Display "data" items in the Client navigation tree**: Select this check box, if you want individual items to be displayed on the MSClient Navigation Tree upon double-clicking. For example, if this option is selected, and you click **Saved Views**, the **Saved View** groups will appear on the Navigation Tree, allowing direct navigation to the **Saved View** group you select.
- **Expand Administrator Tree**: Select this check box, if you want the MSAdmin Navigation Tree to be expanded to second-level navigation options by default.
- **Expand Client Tree**: Select this check box, if you want the MSClient Navigation Tree to be expanded to second-level navigation options by default. This action is available to users, only if they have selected **Tree-based** navigation on the **Site Style** tab under **Preferences** in MSClient.
- **Enable text download in Report View**: Enables users to download text copies of a selected report.
- **Enable PDF download in Report View**: Enables users to download PDF copies of a selected report.
- **Wrap text in PDF Export on Dynamic View**: Select this check box, if you want the text to wrap to a new line, when the edge of a page is reached.
- **Allow using Latest Reports on search**: If you select this check box, MSAdmin and MSClient users will be able to include in their searches only those reports that were added to the system during the last filing.
- **Display dynamic users entitlement report**: Select this check box to add Dynamic User Entitlements Report to Transaction Reports.
- **Allow Mail sending on Client**: Select this check box to allow using Mail sending on client app.
- **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.

Offline Result Settings

- **Offline result expiration in days**: Specify the number of days, on the expiry of which offline results are deleted.

Security Settings

- Permission Control
  - **Allow SubAdmin privilege**: Select this check box if you want to make the **SubAdmin** privilege available, when adding a user, group or role.
  - **Allow SaaSAdmin privilege**: Select this check box, if you want to make the **SaaSAdmin** privilege available, when adding a user, group or role.
  - **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 Client**: Specify the maximum number of attempts to log in using the wrong username or password on MSClient.

Storage Settings

- **Partition size**: Specify the partition size in days. After the expiration of this period a new partition will be created in the database.
- **Maximum files count in SubVolume**: Specify the maximum number of files allowed in the subvolume.
- **Maximum number of simultaneous full-text searches**: Specify the maximum number of users allowed to simultaneously perform full-text search.
- **Number of partitions used in keyword lookup**: Specify the number of partitions to be included in the list of available index values.
- **Document AutoName for failed reports**: Define the default auto-naming scheme to be used for naming documents in case of system errors.
- **Report AutoName for failed reports**: Define the default auto-naming scheme to be used for naming reports in case of system errors.
- **Enable padding in Autoname**: Select this check box, if you want different field values in the names of documents and files to be vertically aligned, by inserting space characters at the beginning of their names.
- **Enable cross-filtering by Keywords**: Select this check box, if you want to turn-on cross-filtering by keywords on the search in client.
- **Enable cross-filtering by Security Keywords**: Select this check box, if you want to turn-on cross-filtering by security keywords on the search in client.

- **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.

Storage defaults

- **Volume Path**: Enter the path to the volume or click the **Browse** button to locate it.
- **Location**: Select location for storage volume.
- **Maximum Volume Size**: Enter the maximum size of the volume measured in megabytes or gigabytes depending on the value you select from the list at the right.
- **Maximum Sub-Volume Size**: Enter the maximum size of the sub-volume measured in megabytes or gigabytes depending on the value you select from the list at the right.

Filing Task defaults

- **Filing Task Path**: Specify the default path to a folder containing files to be processed.
- **Location**: Select the name of a server from the list.

> **NOTE** If the required server is not on the list, refer to Locations Management topic to add a new location.

File processing after filing

Select one of the options listed below, which describe actions for filing files after processing.

- Leave input files where they are
- Delete input files
- Move input files to the location: Specify the location.
- Move files causing errors to folder: Select this check box, if you want to move files causing errors to a special folder and specify the path to the folder.
- Automatically start task when a new file appears: Select this check box, if you want the task to start automatically when a new file appears.
- Log performance statistics: Select this check box, if you want to collect logging information about filing tasks.

0k Input Files processing before filing

Select one of the options below, which describe actions for processing input files with zero size before filing. Files may be deleted or moved to other directory.

- **Delete 0k input files**
- **Move 0k input files to the location** (specify folder)

New Document Types Default Values

Set the values to be used in naming new documents and reports.

- **Default auto name**: Set the string to be used in naming new documents. Click the **AutoName** button to open the Auto Name window, from which you can set a combination of auto name macros to use.
- **Default report name**: Set the string to be used in naming new reports. Click the **ReportName** button to open the Report Name window, from which you can set a combination of auto name macros to use.

> **NOTE** The choices available in the Auto Name and Report Name windows are detailed here.

- **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
Home Page	Default home page (or tab), which is the first page to be displayed when

SETTING	DESCRIPTION
	a user logs on to MSClient.
Document Search page	**Document Search** page to be opened, when you click the Document Search tab.
Report Search page	**Report Search** page to be opened, when you click the Report Search tab.
Document Sorting	Default sort order for search results displayed on the **Document List**.
Report Sorting	Default sort order for search results displayed on the **Report List**.
Maximum Search Result Items	The maximum number of documents that can be listed on the **Document List**.
Display all document/report results	Select this check box to display all document/report results on a single page
Allow multiple Document Type selection in Simple Search	Select this check box to allow selecting multiple document types when performing simple search.
Reuse last search query conditions	Select this check box for the system to remember the dates specified for Simple or Advanced Document/Report Search and to display them when you return to the search page.
Show query info in toolbar	Select this check box to enable the query info toolbar when viewing documents/reports.
Close the popup views toolbar on mouse over	Select this check box for the **Views** toolbar to disappear when the mouse pointer is moved away from it.
Direct view on single result on Reports List	Select this check box to enable direct opening of the report view when clicking its name. This is only valid if the user has the rights only for one type of report view.
Excel Output	**Use XLSX Excel spreadsheet format.** Select this check box to use the XLSX format for Excel output. **NOTE:** This format requires Office 2007 installed on the client computer **Request customize dialog**. Select this check box to customize properties before data export.
Document List Settings	**Count document list items** Specify a numeric value as the default quantity of document list items.
Report List Settings	**Count report list rows**: Specify a numeric value as the default quantity of report list items
Report View	**Count pages**

SETTING	DESCRIPTION
	The maximum number of documents that can be displayed on the Report View page at a time. **Show greenbar** Select this check box to enable alternate row highlighting when viewing report.
Dynamic View	**Count table rows** The maximum number of rows that can be displayed on the Dynamic View page.
Data/Summary View	**Count table rows** Specify the maximum number of rows that can be displayed on the Data View page. **Count summary rows** Specify the maximum number of rows that can be displayed on the Summary View page. **Apply the style from the model** Select this check box to apply the style from the model to the View page. **Apply data formatting from model** Select this check box to apply data formatting from the model to the View page.
CSV Output	**Column Separator** Specify the delimiter for CSV export from RMSClient and MSClient **Apply delimiter from model** Select this check box to apply the delimiter from the model. **Comma** Select this button to use a comma as the delimiter for the CSV export. **Semicolon** Select this button to use a semicolon as the delimiter for the CSV export. **Tab** Select this button to use a tab as the delimiter for the CSV export. **Pipe** Select this button to use a pipe as the delimiter for the CSV export. **Other** Select this button to specify a preferred delimiter.

SETTING	DESCRIPTION
Locale Settings	**Date Format** Specify a date format for your searches.The following options are available: *mm/dd/yyy* or *dd/mm/yyy*. **Decimal Separator** Select one of the proposed decimal separators: comma or period. **Group Separator** Select one of the proposed group separators: comma, space or period
Site Style	**Tabs-based**: This site style horizontally divides the MSClient interface. The top portion of the tab-based interface contains tabs, which provide access to the sections of the application. The bottom portion of the tab-based interface displays the page that corresponds to the selected tab. **Tree-based**: This site style vertically divides the MSClient interface into two halves or panes. The left pane contains the Navigation Tree, which provides links to the various sections of the application, while the right pane displays the page that corresponds to the selected section.
Groups and Roles	**Available User Groups:** Lists available user groups to be assigned to new user. **Available Roles:** Lists available roles to be assigned to new user. Once selected, the user groups and roles assigned to a new user are listed in the Selected User Groups and Selected Roles panes, respectively.

- **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.
- **Maximum allowed report size for PRF Export (Kb)**: Enter the maximum report size for PRF Export in kilobytes.
- **Maximum allowed attachment size (Kb)**: Enter the maximum allowed attachment size in kilobytes.
- **Count of keywords per page**: Defines the number of keywords per page in a **Keyword** dialog box used for searching keywords. Select a value from the list. The default value is set to 10.

- **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.
- **Report view page count (If empty, user preferences are used)**: Select a value from the list, to define the number of pages in a report view. The default setting is **None**.
- Default date range for documents/reports search (leave empty for empty default date range): In this box enter a default date range for document or report search.
- **Maximum number of Documents/Reports to be returned by query**: In this box enter the maximum number of documents or reports to be returned by query.
- **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.
- **Optional columns visibility**: Choose which document properties to display or hide in the documents/reports list.
  - Document Type
  - Document Date
  - State
  - Annotated
  - Native
  - Original
  - Premined
  - Offline
  - Report Size
- **Report Warehouse Home Page Settings**: Choose which pane to display or hide in the MSClient home page.
  - News
  - Work List
  - Favorite Searches
  - Offline Tasks

NOTE When no pane is selected, the MSClient home page will be empty.

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 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).
- **PDF - Annotation Footer Size**: Specify the margin space size for annotations (in inches).
- **PDF - Annotation Font Size**: Specify font size for annotations printed in a footer.
- **PDF - PDF Split Size For Production**: Specify the number of pages to be concatenated to produce a final result.
- **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. A 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.

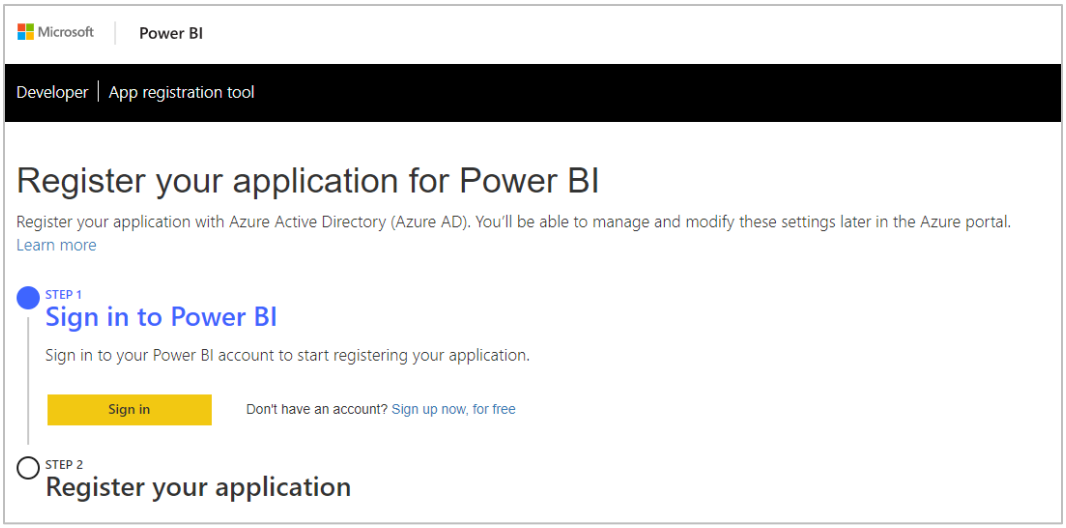
- **Host Name**: It is the full name of the machine on which Automator is installed.
- **Port**: It is the value of the port on which Automator is installed.
- **Redirect URI**: It is the URI to which Power BI directs after authenticating the user. The field is automatically filled after supplying the 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

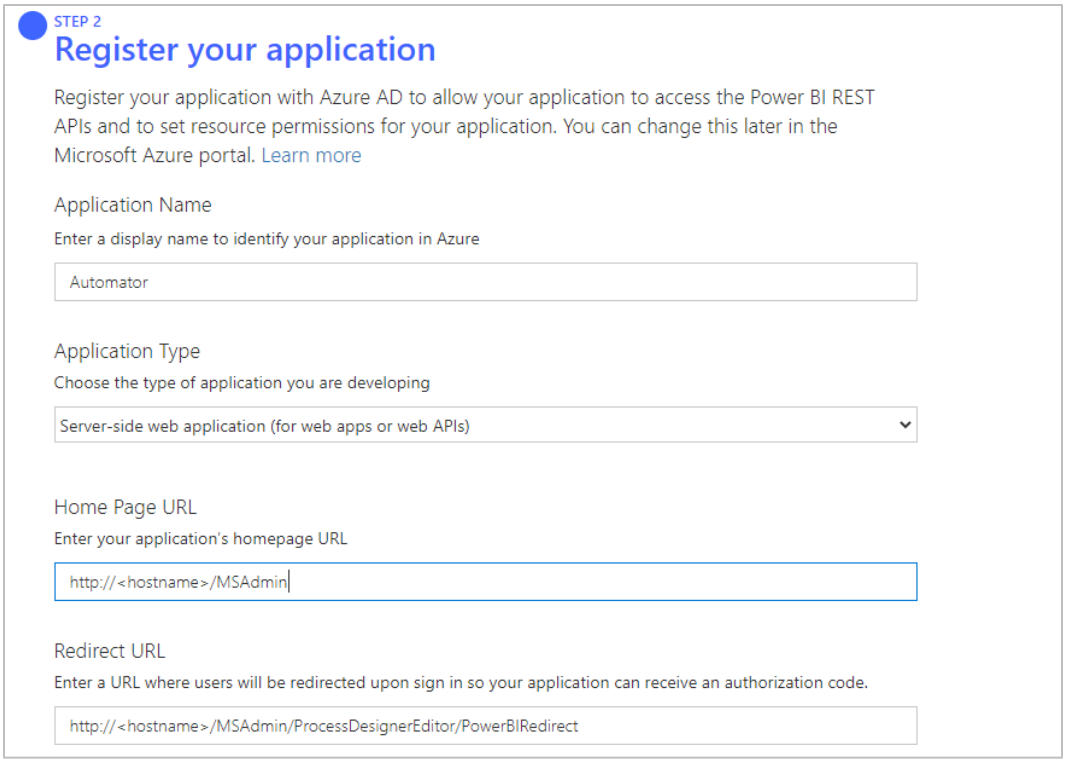
1. Access Power BI's Registration Tool at https://dev.powerbi.com/apps.



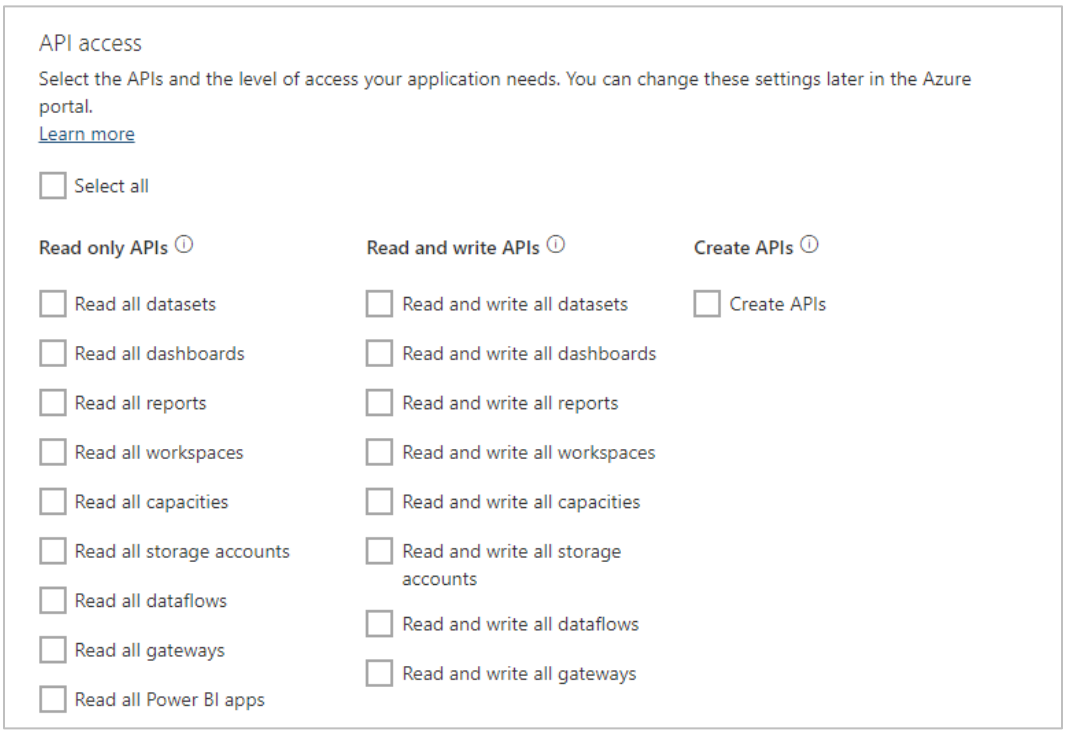
NOTE The registration process in Power BI requires an Azure Active directory, an organizational user, and a Power BI service account. To sign up for an Azure subscription or to log in using an account with such subscription, click here.

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.

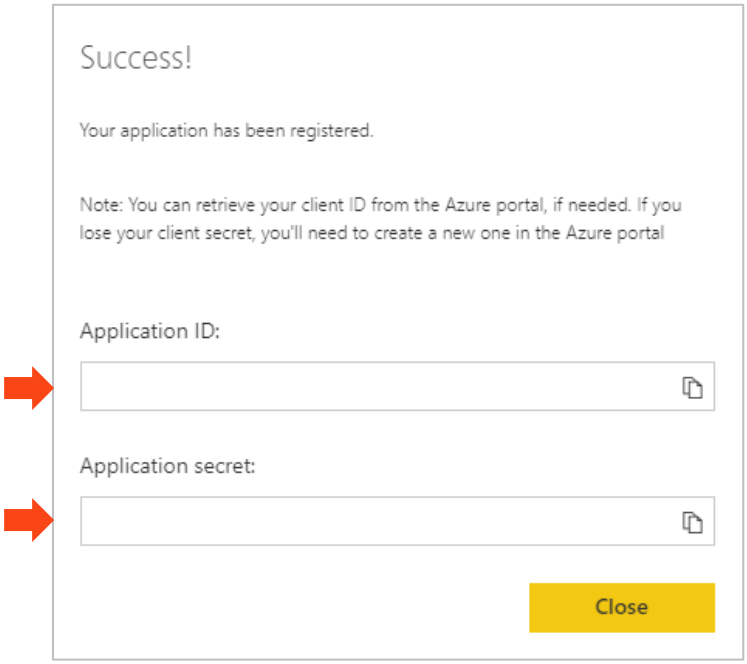


4. Select the APIs you need as well as the level of access required. For details about permissions in Power BI, click here.



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.



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 you 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.
   - **Copy\Move 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).
- **Model Dependencies**: The named path is available to model dependencies (to replace path).
- **Monitoring**: The named path is available to monitoring path in Standard/Visual Processes.
- **Move Disposition**: The named path is available to job log disposition.
- **Preprocess Converter**: The named path is available to preprocess converter item (for filing task).
- **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).
- **Storage Volume**: The named path is available to the storage volume.
- **Virtual Path**. The named path is available to virtual paths.

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.

- **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.

VIRTUAL PATHS

All virtual paths in the system are listed on the Virtual Paths page. To display this page, click **System Settings** on the Navigation Tree, and select **Virtual Paths**.

The list shows:

- Virtual Path Name
- Virtual Path Location
- File Path

Use this page to add, edit, or remove virtual paths from the list.

Adding a Virtual Path

To add a virtual path

1. On the Navigation Tree, click **System Settings**, and select **Virtual Paths**.
2. Click on the **Virtual Paths** page. The **Virtual Path** page appears.
3. Enter the path name in the **Virtual Path Name** field.
4. Specify the CORBA name of the server in the **Location** field, which you can select from the list.
5. Specify the path to a folder containing files to be processed in the **File Path** field or click the **Browse** button to navigate to it.
6. To save the new virtual path, click **Save**.
   The system saves the modified virtual path and displays the updated **Virtual Paths** page, if the operation is completed successfully.
7. To reset the recent changes, click the **Clear** button.
8. To close the **Virtual Path** page without saving, click **Cancel**. Keep in mind that no changes will be saved.

Editing a Virtual Path

To edit a virtual path

1. On the Navigation Tree, click **System Settings**, and select **Virtual Paths**.
2. Click the name of the virtual path you want to edit or click anywhere on the row, and then click . The **Virtual Path** page appears.
3. Make the necessary changes. For more information on how to change its items, see Adding a Virtual Path.
4. To save the changes, click **Save**.
   The system saves the modified virtual path and displays the updated **Virtual Paths** page, if the operation is completed successfully.
5. To reset the recent changes, click the **Clear** button.
6. To close the **Virtual Path** page without saving, click **Cancel**. Keep in mind that no changes will be saved.

Deleting a Virtual Path

To delete a virtual path

1. On the Navigation Tree, click **System Settings**, and select **Virtual Paths**.
2. Do one of the following:
   - Select the virtual path you want to delete, by clicking anywhere on the row, and then click .
   - Click at the end of the row with the virtual path you want to delete.

   A dialog box appears.
3. Click **Delete,** to delete the virtual path 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.

PREPROCESS CONVERTERS

Third party utilities are used by the system for processing files with foreign, non-standard formats.

These utilities are installed separately by the deployment team.

To add a third-party utility

1. Click **System Settings** on the Navigation Tree, and select **Preprocess Converters**.
2. Click + on the **Preprocess Converters** page.
3. Enter the converter name in the **Name** field.
4. Enter the path to the utility executable file or click the **Browse** button to locate it.
5. Enter command parameters in the **Command Parameters** field.

To delete a third-party preprocess converter

1. Select the converter you want to delete, by clicking anywhere on the row, and then click ✖.
2. Click Delete button to confirm deletion.

NOTE Utilities can be removed from the system only when they are not being used in any Filing Tasks.

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, Filing Processes, Maintenance Processes, Offline Processes, Filing Notifications, and Search Notifications.

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
   - Filing Processes
   - Maintenance Processes
   - Offline Processes
   - Filing Notifications
   - Search Notifications
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 the 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.
- Annotation reports provide information about annotations to reports.
- Removed Reports provide information about reports that were removed from **Read-Only** volumes.

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.
- Storage Groups: Lists the existing storage groups, their assigned volumes, their location, etc.
- Document Groups: Lists the existing document groups, their assigned document type(s), their encoding, their index and report models, etc.
- Index Fields: Lists the existing index fields, and provides their ID numbers, type and length limit.
- 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.
- Maintenance Tasks: Lists the existing maintenance tasks, their last start and finish dates, etc.
- Filing Tasks: Lists the existing filing tasks, their last start and finish dates, etc.
- Security: Lists user's security attributes.
- Model Indexes: Reports on problems where mapping of Monarch model fields does not correspond to index fields due to model changes subsequent to document type configuration.
- Document Type Security: Lists all existing users, groups and roles the document type is available for.
- 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.

Storage Groups Report

A Storage Groups report is one of the configuration reports provided by Monarch Server. This report includes all existing storage groups, and provides the following information about them:

- The maximum sub-volume size specified for the groups
- The number of sub-volumes assigned to the groups
- The names of the volumes assigned to the groups
- The location of the volumes (for example, the name of a server, on which they reside)
- The paths of the volumes
- The current size of the volumes

To view and print a Storage Groups configuration report

1. On the Navigation Tree, click **System Reports** and select **Configuration**.
2. Click the **Storage Groups** link. The Storage 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.

Document Groups Report

A Document Groups report is one of the configuration reports provided by Monarch Server. This report includes all existing document groups, and provides the following information about them:

- Document types assigned to each document group
- Encoding formats (e.g., ASCII or Windows-1251) of the document types
- Autoname strings for each document type
- Index and report models for each document type
- Storage group, in which the document types are located
- Report name
- List of assigned indexes

To view and print a Document Groups configuration report

1. On the Navigation Tree, click **System Reports** and select **Configuration**.
2. Click the **Document Groups** link. The Document 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.

Index Fields Report

A Index Fields report is one of the configuration reports provided by Monarch Server. This report includes all existing index fields, and provides the following information about them:

- Names of index fields
- Type of each index field (that is, string, number or date)
- Limit (that is, the maximum length) for each string or number index field

To view and print an Index Fields configuration report

1. On the Navigation Tree, click **System Reports** and select **Configuration**.
2. Click the **Index Fields** link. The Index Fields 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.

Maintenance Tasks Report

A Maintenance Tasks report is one of the configuration reports provided by Monarch Server. This report includes all existing maintenance processes and their assigned maintenance tasks, and provides the following information about them:

- Names of maintenance processes
- Execution frequency of each maintenance process
- Maintenance task(s) assigned to each process
- Type of each maintenance task (for example, a **Delete** task, a **Move** task, and so on)
- Criteria for document selection (for example, a **From...To** date range, the count of documents, the document age in days, and so on) for each maintenance task
- Document types assigned to each maintenance task

To view and print a Maintenance Tasks configuration report

1. On the Navigation Tree, click **System Reports** and select **Configuration**.
2. Click the **Maintenance Tasks** link. The Maintenance 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.

Filing Tasks Report

A Filing Task report is one of the configuration reports provided by Monarch Server. This report includes all existing filing processes and their assigned filing tasks, and provides the following information about them:

- Names of filing processes
- Execution frequency of each filing process
- Filing task(s) assigned to each process
- File name mask of each filing task
- Location (that is, the server) to which reports are filed
- Post-processing actions (for example, leave, move, etc.)
- Document types assigned to each filing task
- Default indexes (if specified) of each filing task

To view and print a Filing Tasks configuration report

1. On the Navigation Tree, click **System Reports** and select **Configuration**.
2. Click the **Filing Tasks** link. The Filing 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.

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.

Model Indexes Configuration Report

A Model Indexes report is one of the configuration reports provided by Monarch Server. This report lists all cases when mapping of Monarch Model fields does not correspond to Index Fields due to model changes subsequent to Document Type configuration.

If an exception is encountered, the report will display it in the following format:

```
ID NAME

-------------------------------

101 Document Type Sample

-------------------------------

Passenger field is mapped for Document Type Sample, but is not existing in current mod
el.

--------------------------------
```

To view and print a Security configuration report

1. On the Navigation Tree, click **System Reports** and select **Configuration**.
2. Click the **Model Indexes** link. The Model Indexes 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.

Document Type Security Report

A Document Type Security report is one of the configuration reports provided by Monarch Server. This report includes all existing users, groups and roles of the document type, and provides the following information about them:

- Document type name and ID
- List of users the document type is available for
- List of groups the document type is available for
- List of roles the document type is available for

To view and print a Document Type Security configuration report

1. On the Navigation Tree, click **System Reports** and select **Configuration**.
2. Click the **Document Type Security** link. The Document Type Security 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:

- Document Actions — Lists the time, date, user, etc., of document actions.
- Report Actions — Lists the time, date, user, etc., of report actions.
- Search Actions — Lists the time, date, user, etc., of search actions.
- Security — Lists the time, user name, and log on/log off information.
- Filing — Lists the time, user name and filing process information.
- Maintenance — Lists the time, user name, and maintenance task information.
- System — Lists system status information (e.g., whether the e-mail system has been started, the sub-volumes have been mounted, etc.).
- Reports Archived — Lists information about each report stored into Monarch Server

- 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.
- Deleted Documents — Lists detailed information about deleted documents.
- Report Variance — Lists information about read-only volume's storage changes (i.e. non-existence or inequality of several zip storages).
- Deleted Filing Results — Lists information deleted filing results.
- Audit Entity Actions — Lists information about audit entity actions.
- Custodian Schedules Logs — Lists information about custodian schedules events.
- Dynamic Users Entitlement — List information about dynamic users entitlement (available if the corresponding check box is selected in System Preferences)
- 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.

Document Actions Report

A Document Actions Report is one of transaction reports provided by Monarch Server. This report provides the following information:

- Time when documents were viewed
- Name(s) of user(s) who viewed the documents
- Document(s) viewed by user(s)

To view and print a Document Actions transaction report

1. On the Navigation Tree, click System Reports and select Transaction.
2. Click the **Document Actions** link. The Document Actions Transaction Report page is displayed.
3. In the **From ... To** fields specify a time period, for which you want to view the document actions. Click to open the Calendar window, and select the date.

> **NOTE** If you do not provide a date or time period, all document 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 document actions for all users, select **All Users**.)

5. From the **Document Groups** list, select a document group, to which the desired document type(s) is assigned.
6. From the **Document Types** pane, drag appropriate items to the **Assigned Document Types** pane.
7. If you want to view actions for one document only, click the **Show Documents** button under the panes. The **Documents** list will display all documents of the selected document types. You can select one document to be included in the report.

> **NOTE** By default the **All Documents** option is set in the **Documents** list.

8. Click **View**. The Document Actions transaction report is displayed.
9. 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.

Report Actions

A Report Actions Report is one of transaction reports provided by Monarch Server. This report provides the following information:

- Time when reports were viewed
- Name(s) of user(s) who viewed the reports
- Reports(s) viewed by user(s)

To view and print a Report Actions transaction report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **Report Actions** link. The Report Actions Transaction Report page is displayed.
3. In the **From ... To** fields specify a time period, for which you want to view the report actions. Click to open the Calendar window, and select the date.

> **NOTE** If you do not provide a date or time period, all report 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 report actions for all users, select **All Users**.)
5. From the **Document Groups** list, select a document group, to which the desired document type(s) is assigned.
6. From the **Document Types** pane, drag appropriate items to the **Assigned Document Types** pane.
7. If you want to view actions for one report only, click the **Show Documents** button under the panes. The **Reports** list will display all reports of the selected document types. You can select one report.

NOTE By default the **All reports** option is set in the **Reports** list.

8. Click **View**. The Report Actions transaction report is displayed.
9. 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.

Search Actions Report

A Search Actions Report is one of transaction reports provided by Monarch Server. This report provides the following information:

- Time when the search was performed
- Name of a user(s) who performed the search(es)
- Search(es) performed by a user(s)

To view and print a Search Actions transaction report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **Search Actions** link. The Search Actions Transaction 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 **Search Groups** list, select a search group. The searches assigned to it appear in the **Search** box.
6. Select a search in the **Search** box, then click **View**. The Search Actions Transaction Report is displayed.
7. 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 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.

Filing Report

A Filing Report is one of transaction reports provided by Monarch Server. This report provides the following information:

- Date and time when each filing process and task began and finished running.
- Names of filing processes.
- Filing task(s) assigned to each process.
- Error message for any task or process that failed to run.

To view and print a Filing transaction report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **Filing** link. The Filing Report page is displayed.
3. In the **From ... To** fields specify a time period, for which you want to view the filing actions. Click to open the Calendar window, and select the date.

NOTE If you do not provide a date or time period, all filing actions will be displayed.

4. Click **View**. The Filing 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.

Maintenance Report

A Maintenance Report is one of transaction reports provided by Monarch Server. This report provides the following information:

- Date and time when each maintenance process and task began and finished running.
- Names of maintenance processes.
- Maintenance task(s) assigned to each process.
- Error message for any maintenance task or process that failed to run.

To view and print a Maintenance transaction report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **Maintenance** link. The Maintenance Report page is displayed.
3. In the **From ... To** fields specify a time period, for which you want to view the maintenance actions. Click to open the Calendar window, and select the date.

NOTE If you do not provide a date or time period, all maintenance actions will be displayed.

4. Click **View**. The Maintenance 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.

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.

Reports Archived

Reports Archived Report is one of the transaction reports provided by Monarch Server. This report provides the following information:

- Date of report filing
- Report name
- Number of report files processed in the task
- Total number of documents contained in the files
- Maximum number of lines per document
- Total number of pages filed

To view and print a Reports Archived transaction report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **Reports Archived** link. The Reports Archived 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 Reports Archived 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.

Deleted Documents Report

A Deleted Documents Report is one of transaction reports provided by Monarch Server. This report provides the following information:

- ID of a deleted document
- Detailed information about the document (volume name, storage group name, user name, report name, document date, date and time of deletion, path).

To view and print a Deleted Documents transaction report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **Deleted Documents** link. The Deleted Documents page is displayed.
3. In the **From ... To** fields specify a time period, for which you want to view deleted documents. Click to open the Calendar window, and select the date.
4. From the **User** list select a user, to view documents deleted by them.
5. From the **Document Groups** list, select a document group, containing deleted documents that you want to view.
6. From the **Assigned Document Types** list, select an appropriate document type.
7. Click **View**. The Deleted Documents 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.

Report Variance

A Report Variance Report is one of transaction reports provided by Monarch Server. This report provides the following information:

- ID of a missing or corrupted file
- Variance Type
- Detailed information about the file (report date, filling date, document type, report name, file size in the database and file size the storage).

To view and print a Report Variance transaction report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **Report Variance** link. The Report Variance page is displayed.
3. In the **From ... To** fields specify a time period, for which you want to view report variance actions. Click to open the Calendar window, and select the date.

> **NOTE** If you do not provide a date or time period, all report variance actions will be displayed.

4. From the **Document Groups** list select a document group.
5. From the **Document Types** pane drag an appropriate item(s) to the **Assigned Document Types** pane.
6. Click **View**. The Report Variance Report is displayed.
7. 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 Filing Results Report

A Deleted Filing Results Report is one of transaction reports provided by Monarch Server. This report provides the following information:

- Date and time when filing results were deleted
- Name of a user who deleted filing results
- Name of a filing task

To view and print a Deleted Filing Results transaction report

1. On the Navigation Tree, click **System Reports** and select **Transaction**.
2. Click the **Deleted Filing Results** link. The Deleted Filing Results 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 Filing Results 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.

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.

ANNOTATION REPORTS

Annotation reports provide information about all annotations to reports.

This report provides the following information:

- Time when annotations were created
- Annotation level (report, page or selection)
- Name(s) of user(s) who created annotations
- Text of annotations created by a user(s)

To view and print an Annotation report

1. On the Navigation Tree, click **System Reports** and select **Annotations**.
2. In the **Annotations: From ... To** fields specify a time period, for which you want to view annotations. Click to open the Calendar window, and select the date.

> **NOTE** If you do not provide a date or date range, all annotations for the specified user(s) will be included in the report.

3. In the **Document date: From ... To** fields specify a time period for the document dates to be included in the report. Click to open the Calendar window, and select the date.

> **NOTE** Date range including less than 90 days is required for document date.

4. From the **User** list, select a user whose annotations you want to view (to view annotations of all users, select the **All Users** option).
5. From the **Document Groups** list select a document group, to which the desired document type(s) is assigned.
6. From the **Document Types** pane drag an appropriate item(s) to the **Assigned Document Types** pane.
7. If you want to view annotations for one report only, click the **Show Documents** button under the panes. The **Reports** list will display all reports of the selected document types. You can select one report to be included in the report.

> **NOTE** By default, the All reports option is set in the Reports list.

8. Click **View**. The Annotation Report is displayed.
9. 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.

Annotations

An annotation is a portion of a document to which a text message has been attached. When a document that contains an annotation is viewed on the Report View page within MSClient, the annotated portion of the document is highlighted in yellow.

When MSClient users add an annotation to a document, they specify the role members, group members and users, to whom they want the annotation to be visible. Any user who subsequently views that document, and to whom the annotation has been assigned, will be able to view the text message.

NOTE MSClient users can only add annotations to documents if an administrator has granted them the **Create Annotation** privilege.

REMOVED REPORTS

Monarch Server provides possibility to view the following information about reports that were removed from **Read-Only** volumes:

- Availability of a **First Copy** and a **Second Copy** at the moment of deletion
- File name
- Filing date
- Document date
- File size
- Report name

To view removed reports

1. On the Navigation Tree, click **System Reports** and select **Removed Reports**.
2. In the **Document date: From ... To** fields specify a time period for the document dates to be included in the report. Click to open the Calendar window, and select the date.
3. From the **Volumes** list, select a read-only volume, from which reports were removed.
4. From the **Document Types** pane, select a document type that contained removed reports.
5. Click **View**.
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

ACTIVITY MONITOR

Monarch Server provides the possibility to monitor current filing and maintenance activities, performed in the system.

To access the activity monitor, on the **Navigation Tree**, click **System Reports**, and select **Activity Monitor**.

To display the list of activities occurring in the system

1. From the Job Type list, select Filing Process, Auto start Filing Process, or Maintenance Process.
2. From the **Time Period** list, select one of the following options:
   - **Now** (to show processes occurring at the current moment)
   - **Last Hour** (to show processes that occurred during the last hour)
   - **Last 24 Hours** (to show processes that occurred during the last 24 hours)
   - **Nearest** (to show processes that will occur in the near future)

The list displays the following information:

- Name of a process
- Start time
- End time
- **Information** (click a plus sign to display additional information about a process)
- **Status** (Not Started, In Progress or Completed)

NOTE	The list auto-refreshes every 60 seconds. You can refresh it manually by clicking **Refresh**.

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.

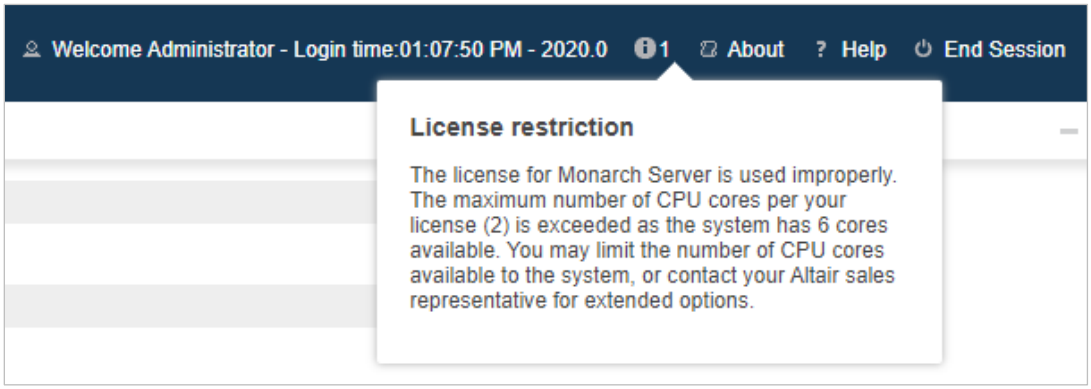
Before Contacting Technical Support

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. There 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

v2021.1.1/20220512