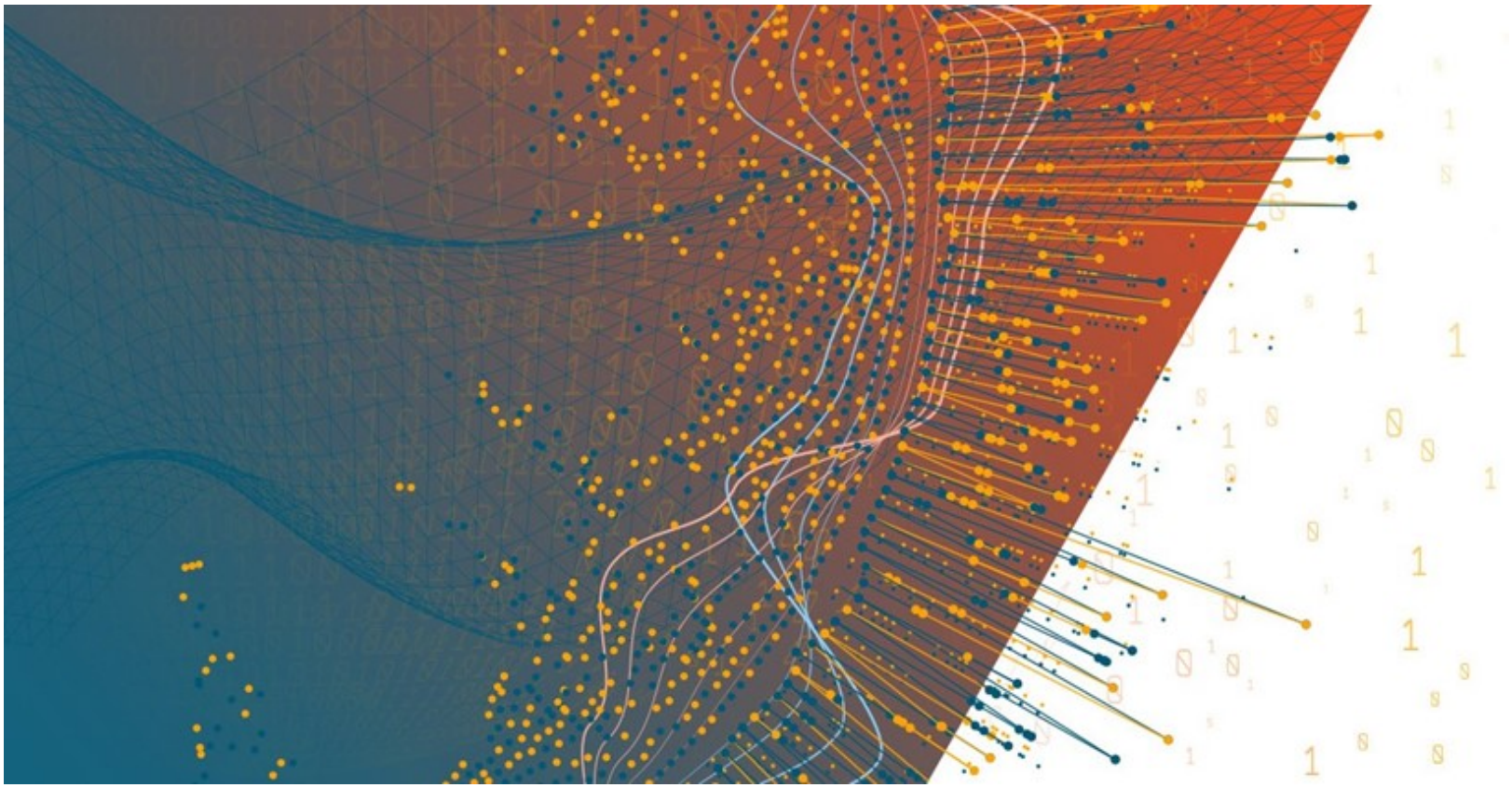


Altair® Monarch Server™ 2025.0

Automator - Administrator's Guide

TABLE OF CONTENTS

Introduction **9**

About Monarch Server Automator Edition **10**

Monarch Server Automator Edition Workflow **11**

Launching Monarch Server Web Administrator **11**

Navigation Tree **14**

Functional Buttons **17**

Page Navigation **17**

Using the Calendar **18**

Report Warehouse **20**

Maintenance Management **20**

Log Clearance Tasks 20

Adding a Log Clearance Task 21

Editing a Log Clearance Task 22

Deleting a Log Clearance Task 22

Running a Log Clearance Task 22

Maintenance Processes 23

Adding a Maintenance Process 23

Execution Frequency 24

Cron Functionality 26

Running a Maintenance Process 35

Editing a Maintenance Process 36

Deleting a Maintenance Process 36

Automator **37**

Server Status **37**

Standard Processes **40**

Processes 40

Adding a Process 42

Editing a Standard Process 43

Specifying General Settings 44

Specifying Advanced Settings 44

Specifying Job Log Disposition Settings for a Process 47

Specifying Input File Monitoring Settings for a Process 48

Specifying Scripts for a Process 50

Scheduling a Process 50

Adding a Schedule 51

Editing a Schedule 51
Deleting a Schedule 52
Assigning Process Access Rights 52
Deleting a Process 53
Enabling/Disabling Monitoring of Standard Processes 53
Enabling/Disabling Scheduled Standard Processes 54
Running a Process 54
Importing a Standard Process 55
Exporting a Process 56
Process Handling Scenarios 56
Schedules 60
Execution Frequency 60
Cron Functionality 26
Jobs 72
Cancelling a Job 73
Deleting a Job 74
Distributions 74
Adding a Distribution 75
Editing a Distribution 75
Adding a Distribution Action 76
Translate Distribution Action 76
Move Distribution Action 77
RSS Distribution Action 78
Log Distribution Action 79
Copy Distribution Action 80
Email Distribution Action 81
Delete Distribution Action 82
Editing a Distribution Action 82
Deleting a Distribution Action 83
Deleting a Distribution 83
Alerts 83
Adding an Alert 84
Editing an Alert 85
Adding an Alert Action 85
Email Alert Action 86
RSS Alert Action 87
Log Alert Action 88

Editing an Alert Action ... 88
Deleting an Alert Action ... 89
Deleting an Alert ... 89
Global Environment Scripts ... 89
Project Editor ... 90
Creating Project Files ... 90
Specifying General Settings for a Project File ... 91
Viewing Project Information ... 91
Specifying Model Settings for a Project File ... 92
Verifying Project Reports ... 92
Viewing Model Information ... 94
Specifying Input Settings for a Project File ... 95
Specifying Export Settings for a Project File ... 97
General Screen ... 99
Table View Screen ... 100
Output File Screen ... 100
Summary View Screen ... 102
PDF Security Screen ... 103
PRF Info Screen ... 104
Tableau Server screen ... 105
OleDb Info Screen ... 105
Table Info Screen ... 106
Column Info Screen ... 107
Excel Advanced Features Screen ... 108
Specifying Default Export File Types ... 109
Specifying Prerequisite Settings for a Project File ... 109
Specifying Script Settings for a Project File ... 110
Specifying Login Credentials ... 111
Visual Processes ... 112
Processes ... 112
Adding a Visual Process ... 114
Editing a Visual Process ... 114
Deleting a Process ... 115
Enabling/Disabling Monitoring of Visual Processes ... 115
Enabling/Disabling Scheduled Visual Processes ... 116
Visual Process Designer ... 116
Input Elements ... 127

Model Elements ... 131
DataPrep Elements ... 133
Export Elements ... 152
Prerequisites Elements ... 163
Misc Elements ... 168
Output Elements ... 179
Process Designer Scripting API ... 180
Importing a Visual Process ... 185
Exporting a Visual Process ... 186
Migrating a Standard Process ... 187
Schedules ... 187
Process Templates ... 188
Process Logs ... 188
Alerts ... 190
Adding an Alert ... 191
Editing an Alert ... 193
Deleting an Alert ... 193
Scripts ... 194
Script Editor ... 194
PreExport Script ... 195
PostExport Script ... 196
PreProcess Script ... 196
PostProcess Script ... 196
Global Declarations Script for Standard Processes ... 197
Global Declarations Script for Visual Processes ... 197
References and Imports ... 197
Passwords ... 198
Inserting a Script for Standard Processes ... 198
Inserting a Script for Visual Processes ... 201
Testing a Script ... 203
Guidelines for Valid and Invalid Characters ... 204
Connection Definitions ... 206
Using the Naming Macros ... 207
Offset Expressions in Macros ... 212
When a Job Fails to Run ... 214
Runtime Fields ... 214
Project Inputs via Manifest ... 216

Project Outputs via Manifest **218**
Standard Processes with Manifest Monitoring **222**
Visual Processes with Manifest Monitoring **228**
XSD for Standard Processes **233**
XSD for Visual Processes **237**
Server Library **258**
Folders **258**
Adding a Folder 258
Editing a Folder 260
Assigning Folder Access Rights 260
Deleting a Folder 261
Projects **262**
Adding a Project File 262
Editing a Project File 263
Assigning Project File Access Rights 263
Project File Revisions 264
Opening a File in the Project Editor 265
Deleting a Project File 265
Changing a Project File State 266
Models **266**
Adding a Model 266
Editing a Model 267
Assigning Model Access Rights 269
Deleting a Model 270
Changing Model State 270
Workspaces **270**
Adding a Workspace 271
Editing a Workspace 273
Assigning Workspace Access Rights 274
Deleting a Workspace 275
Changing Workspace State 275
Change Lists **276**
Adding a Change List 276
Editing a Change List 277
Assigning Change List Access Rights 278
Deleting a Change List 279
Changing Change List State 279

Bulk Edits **280**
Rights and Privileges **282**
Assigning Privileges **282**
Automator Administrator 283
Automator User 283
Automator Light User 284
Assigning Access Rights **284**
Setting Security Provider **285**
Roles **286**
Adding a Role 286
Editing a Role 288
Deleting a Role 289
User Groups **289**
Adding a User Group 290
Importing a User Group 291
Editing a User Group 292
Deleting a User Group 293
Users **293**
Adding a User 293
Changing User Password 295
Editing a User 295
Deleting a User 297
System Settings **298**
System Preferences **298**
Application Settings 300
About Information 302
Time Intervals 302
New User Defaults 303
Global User Settings 303
Automator Settings 304
General Settings 304
Custodian Schedule 305
Adding a Custodian Schedule 306
Editing a Custodian Schedule 306
Deleting a Custodian Schedule 307
Automator Contacts 307
Automator LDAP Settings 308

Default Job Log Disposition 308
Server Integration 309
Input 320
Locations Management 320
Named Paths 321
Exception Dates 323
Locale Management 324
Bulk Schedules Edits 326
System Reports 328
Configuration Reports 328
Product License Report 328
Roles Report 330
User Groups Report 331
Users Report 332
Security Configuration Report 334
Automator Process Dependencies 335
Transaction Reports 336
Security Report 337
Added Users Report 339
Deleted/Suspended Users Report 340
Users Not Logged In Report 341
Audit Entity Actions Report 343
Custodian Schedules Logs Report 344
Automator Distribution Audit 346
Converting a Report to Another Format 347
Active Users 348
Technical Support 349
Glossary 351

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 Report Mining Server (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.

NOTE: This guide also describes a few functions of Report Warehouse that are accessible alongside Automator. For further information about Report Warehouse, click here.

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

- Automator
- Automator Workflow
- Launching MSAdmin
- Navigation Tree
- Functional buttons
- Page Navigation
- Using the Calendar

NOTE: To view the latest updates in the Monarch Server documentation, visit https://help.altair.com/.

About Monarch Server Automator Edition

Monarch Server Automator 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 Automator 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 Automator Edition also allows you to export files to a Microsoft Office SharePoint Server, FTP site, email destination, or distribution list. In short, Monarch Server Automator Edition lets you extract, transform, move and load data faster and more easily than previously possible.

Monarch Server Automator 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 Automator Edition is a perfect solution for extracting data from periodic reports on a regular basis. For example, you might instruct Monarch Server Automator 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 Automator Edition to monitor specified folders, so that as soon as a file arrives, the conversion process begins.

Monarch Server Automator 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 Automator 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 Automator Edition Workflow

When using Monarch Server Automator 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 Automator 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.

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 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.

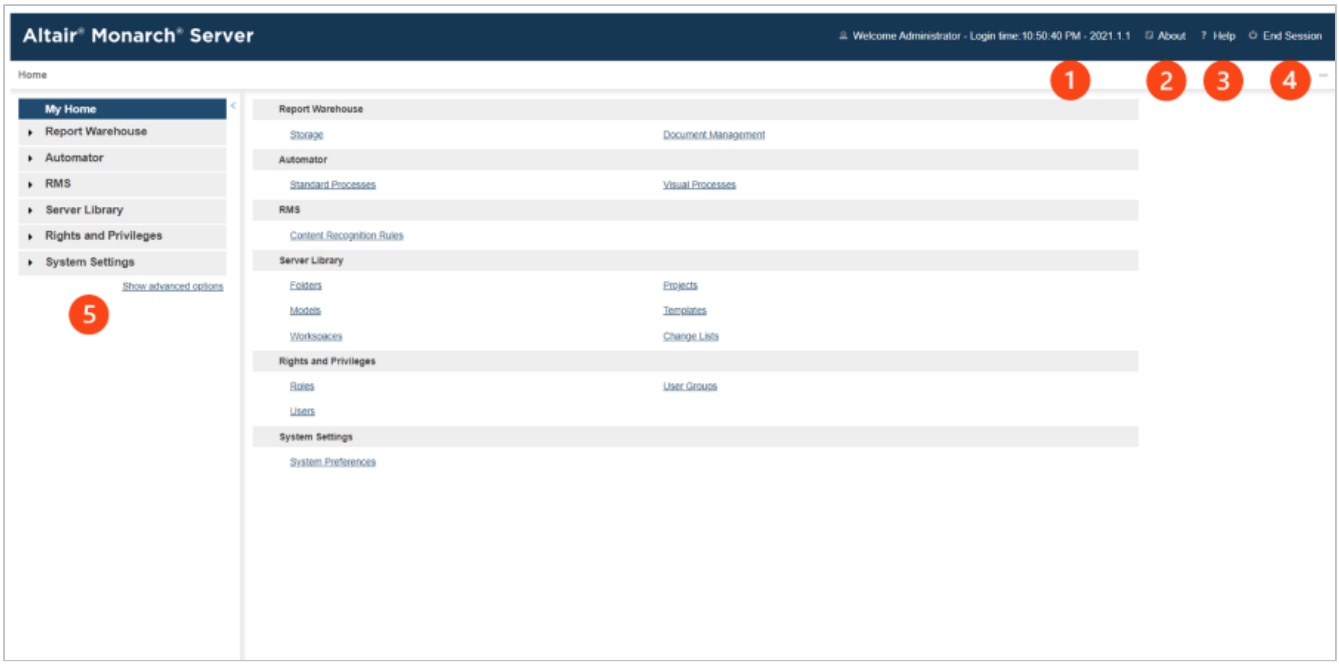
NOTES:

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 2025.0, the SDK and DataPrep versions are as follows: **SDK version**: 25.0.0.37738 **DataPrep version**: 25.0.0.37738
3	**Help** Clicking this button opens a related page in the Administrator's Guide.
4	**End Session**

Panels and Functions	Description
	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 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:

Section Name	Description
My Home	This page is displayed by default, when you log on to MSAdmin. The right pane contains links to all sections of the system.
Report Warehouse	
Maintenance	
• Log Clearance Tasks	This link allows you to manage log clearance tasks. For more information, see Log Clearance Tasks.
• Maintenance Processes	This link allows you to manage processes. For more information, see Maintenance Processes.
Automator	
Server Status	This link allows you to check the current status of the Monarch Server Automator 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.

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.
Workspaces	This link allows you to manage workspaces. For more information, see Workspaces.
Change Lists	This link allows you to manage change lists. For more information, see Change Lists.
Bulk Edits	This link allows you to apply text content bulk editing to Server Library items. For more information, see Bulk Edits.
Rights and Privileges	
Security Provider	This link allows you to define a security provider. For more information, see Setting Security Provider.
Roles	This link allows you to manage roles. For more information, see Roles.
User Groups	This link allows you to manage user groups. For more information, see User Groups.
Users	This link allows you to manage users. For more information, see Users.
System Settings	
System Preferences	This link allows you to set system preferences. For more information, see System Preferences.
Locale Management	This link allows you to manage locales. For more information, see Managing Locales.
Bulk Schedules Edits	This link allows you to apply bulk editing of schedules. For more information, see Bulk Schedules Edits.
System Reports	

Configuration	This link allows you to generate configuration reports. For more information, see Configuration Reports.
Transaction	This link allows you to generate transaction reports. For more information, see Transaction Reports.
Active Users	This link allows you to monitor active users in the system. For more information, see Active Users.

Functional Buttons

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

+ Allows you to add a new item to the list displayed in the right pane.

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

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

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

To remove an item

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

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

Page Navigation

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

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

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

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:

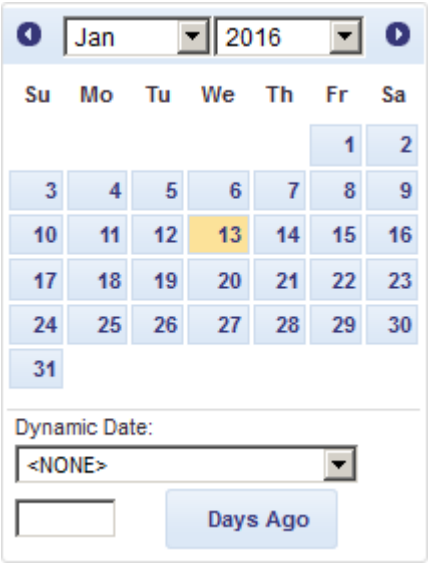
- To navigate to the next page, click .
- To navigate to the last page, click .
- To navigate to the previous page, click .
- To navigate to the first page,click .

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

- Select a month on the **Calendar** using arrows and then select the date.
- 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.
- Set a date as a number of days ago. Enter the corresponding value in the box and click **Days Ago**.
- You can also use the following buttons, placed next to a date field:
    - To decrease the date by one day, click .
    - To enter the current date, click .
    - To increase the date by one day, click .

Report Warehouse

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

The topics in this section provide information about:

- Maintenance Management

Maintenance Management

The topics in this section provide information about

- Log Clearance Tasks
- Maintenance Processes

Log Clearance Tasks

A log clearance task deletes records of a specified type from the Logs, and Schedule Jobs, Pump Job Logs, 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.

- **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 Schedules Jobs and Visual Process (for Automator) 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.

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.

When a maintenance process runs, it executes all 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 and 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.

- **Cron**: Select this option if you want to set a cron schedule.
  - **Expression**: Enter the cron expression and the press the **TAB** key to validate it. The following guidelines are available for cron expressions:
    - The maximum length is 64 characters.
    - The expression should have 5 fields only, as shown in the following format: **<minutes> <hours> <day of the month> <month> <day of the week>** (e.g., 20 20 ? * ?).
    - The "seconds" field is not accepted; the first cron field specified is interpreted as "minutes."

- For the "month" field, the numeric values 1 to 12 are accepted; abbreviations such as JAN and DEC are not allowed.
- For the "day of the week" field, the numeric values 0 to 6 are accepted; abbreviations such as SUN and SAT are not allowed.

NOTE: Detailed documentation for cron, including descriptions and examples, is available here.

- In the **At** check box set the time when the current process will be launched following the **HH:MM** format.

CRON Functionality

This section describes in detail the CRON functionality in Monarch Server. (For French, click here.)

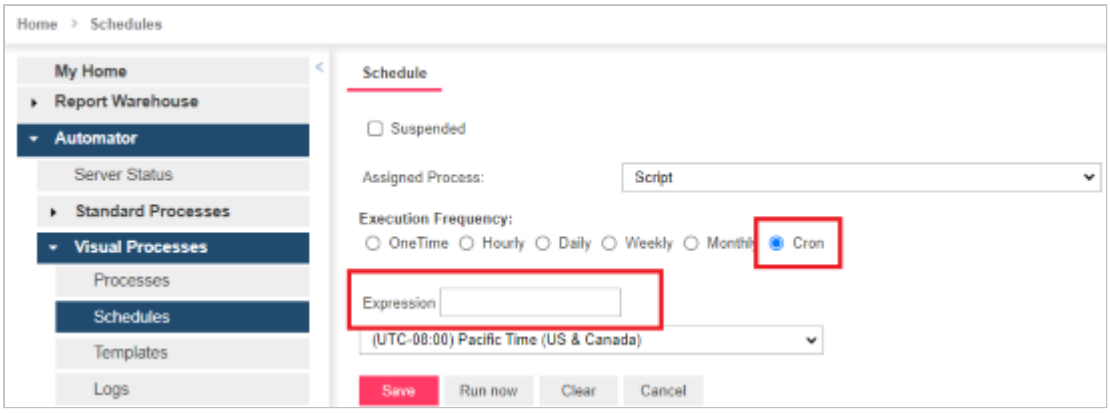
What is CRON?

CRON is a time-based job scheduler that is traditionally used in Unix-like operating systems but has since been inherited by others. It is designed to automatically execute commands or scripts at specific times or intervals. It is commonly used to automate repetitive tasks, such as backups, email notifications, system maintenance, and data processing.

CRON in Monarch Server

Monarch Server has background modules that run continuously and execute scheduled tasks.
Once configured, the modules check for associated task schedules to run as frequently as every minute (assuming correct instance configuration) based on internally stored CRON expressions, which specify the exact times or intervals when tasks should run.

In the Monarch Server scheduling interface (version 2024.1+), a new option allows for the provision of a user-defined CRON string as opposed to using the scheduling interface to define the same via the options presented by (and limited to) the interface options.



When the CRON option is selected, you can either enter or paste the CRON string desired in the Expression box.

Monarch Server CRON Expression Format

CRON expressions consist of five fields that define the schedule, plus the command or script to execute. The standard format is as follows:

```
┌──────────── Minute (0 - 59)
│ ┌──────────── Hour (0 - 23)
│ │ ┌──────────── Day of the month (1 - 31)
│ │ │ ┌──────────── Month (1 - 12)
│ │ │ │ ┌──────────── Day of the week (0 - 7) (Sunday = 0 or 7)
│ │ │ │ │
│ │ │ │ │
* * * * * CRON Expression (five parts only)
```

Monarch Server CRON Expression Symbols

Symbol	Description
*	Wildcard, meaning "every" value in that field

,	Allows specifying multiple values in the same field (e.g., 1,15 for the minute field means 1st and 15th minute)
-	Defines a range of values (e.g., 9-17 for the hours field means from 9 AM to 5 PM)
/	Defines a step value (e.g., */15 for the minutes field means every 15 minutes)

NOTE:

Not all CRON systems support enhanced characters such as **L**, **W**, **#**, or **?**.

Standard CRON (per the Monarch Server implementation) supports basic characters such as *****, **,**, **-**, and **/**.

Advanced characters such as **L**, **W**, **?**, and **#** are typically found in Quartz or other enhanced schedulers and are NOT supported in Monarch Server.

Examples of CRON Expressions

Here are some typical CRON jobs with their schedules:

CRON Expression	**Schedule**
* * * * *	Every minute
0 0 * * *	Every day at midnight
0 0 * * 1	Every Monday at midnight
*/15 9-17 * * 1-5	Every 15 minutes, 9 AM to 5 PM, Monday through Friday

Monarch Server has historically stored user-defined schedules internally using the CRON syntax. However, post-Monarch Server 2024.1, administrators can define schedules using CRON syntax explicitly, thus allowing for more efficient and complicated and customised scheduling definitions.

Generating CRON Expressions

CRON expressions can be created using the following options:

- Copying/pasting of existing strings (assuming they comply with the traditional five-part CRON/CronTab notation used in Monarch Server)
- Deriving expressions directly based on existing CRON knowledge and experience
- Using the samples provided in this document (with potential modification as desired)
- Using online tools and resources that provide other examples or assist in creating new expressions interactively. Some useful examples of these resources are as follows:
    - Crontab.guru - The cron schedule expression generator: a simple CRON expression builder/editor;
    - Cron Expression Generator | Uptimia: allows you to provide a natural language-based requirement which the site will attempt to parse into a CRON string;
    - Cron job editor: multiple cron jobs, calendar view, AWS & Vercel cron support | CronTool: simple CRON editor/debugger with an on-screen calendar to help visualize the subsequent execution event(s); a more advanced version of CronTool allowing for multiple CRON expressions to be “stacked” to see their combined execution events.

NOTE: Some of the CRON schedules will require script to be added as part of the process to enable exclusions and exceptions outside of the scope of the CRON statement or construct limitations.

Benefits of CRON Expressions

Using CRON strings for scheduling offers several advantages.

1. **Precision and Flexibility**
    - **Fine-Grained Scheduling**: CRON strings allow for precise scheduling down to the minute. You can specify tasks to run at exact times (e.g., every hour, specific days, or monthly).
    - **Complex Schedules**: CRON strings can handle both simple and complex scheduling needs, such as "every 15 minutes during business hours." This flexibility is essential for tasks that require specific timing.
    - **Patterns and Recurrence**: With symbols like * (wildcard), / (step), and ranges (-), CRON can define recurring intervals without having to specify each individual time.
2. **Consistency and Reliability**

- **Automated Execution**: Once a CRON job is set up, it can run automatically at the scheduled times without any manual intervention. This reliability ensures that critical tasks can be consistently performed on schedule, assuming a correctly configured Monarch Server instance.
- **Repeatability**: CRON jobs run consistently across time zones, system reboots, and other environmental changes, making them highly dependable for tasks that must occur regularly.

3. **Simplicity and Ease of Use**
   - **Concise Syntax**: CRON strings use a compact, standardized syntax that's easy to understand and customize once learned. A comprehensive list is included later in this document.
   - **Minimal Configuration**: CRON strings require minimal setup and can be easily configured by an administrator in the Monarch Server Admin UI. This simplicity is particularly helpful for system administrators managing multiple tasks or where a task requires multiple schedules.
4. **Versatile Applications Across Systems**
   - **Wide Compatibility**: CRON is available on almost all Unix-like systems, making it a universal scheduling tool across Linux, macOS, and other systems. This portability is especially advantageous in heterogeneous environments.
   - **Suitable for Various Tasks**: CRON can schedule almost any command or script, including a maintenance script, a data backup, or an email notification, making it versatile for many automation needs.
5. **Customizability for Complex and Conditional Scheduling**
   - **Exceptions and Conditional Logic**: By combining CRON with process scripting or checks in the executable code, you can create conditional schedules (e.g., excluding certain dates and adjusting for holidays).

Common CRON Examples

Below is a list of common CRON expressions with descriptions of what they do. These examples can help cover most typical scheduling needs.

For reference, the standard format is

```
┌──────────── Minute (0 - 59)
│ ┌──────────── Hour (0 - 23)
│ │ ┌──────────── Day of the month (1 - 31)
│ │ │ ┌──────────── Month (1 - 12)
│ │ │ │ ┌──────────── Day of the week (0 - 7) (Sunday = 0 or 7)
│ │ │ │ │
│ │ │ │ │
* * * * * CRON Expression (five parts only)
```

CRON Expression	Description
* * * * *	Every minute
*/5 * * * *	Every 5 minutes
0 * * * *	Every hour, on the hour
0 */2 * * *	Every 2 hours, on the hour
0 0 * * *	Every day at midnight
0 9 * * *	Every day at 9:00 AM
0 0 * * 1	Every Monday at midnight
0 0 1 * *	The first day of every month, at midnight
0 0 1 1 *	Every January 1st at midnight
0 0 * * 0	Every Sunday at midnight
*/15 9-17 * * 1-5	Every 15 minutes during business hours (9:00 AM to 5:00 PM) Monday through Friday
0 12 * * 1-5	Every weekday at noon
0 0 1-7 * 0	The first Sunday of every month, at midnight

0 0 */3 * *	Every third day
*/10 0 1 1 *	Every 10 minutes on January 1st, between midnight and 12:59 AM
0 0 25 12 *	Every Christmas Day at midnight
30 14 1 * *	The first day of every month at 2:30 PM
0 8,16 * * *	Every day at 8:00 AM and 4:00 PM
*/5 8-10 * * *	Every 5 minutes, between 8:00 AM and 10:59 AM
0 0 1-7 * 0	The first Sunday of every month
0 6,18 * * *	Every day at 6:00 AM and 6:00 PM
*/30 * * * *	Every 30 minutes
0 22 * * 1-5	Every weekday at 10:00 PM
0 7 * * 1	Every Monday at 7:00 AM
*/2 * * * 3	Every 2 minutes, on Wednesdays
15 14 1 * *	Every month on the 1st, at 2:15 PM
0-4 14 * * *	Every minute starting at 2:00 PM and ending at 2:05 PM daily
5 * * * *	At 5 minutes past every hour
15 10 15 * *	At 10:15 AM on the 15th day of every month
15 10 15 * 1-5	At 10:15 AM every Monday through Friday
0 0 15-21 * 5	At 10:15 AM on the third Friday of every month

CRON Examples with Conditions

Conditional CRON expressions are powerful tools for defining schedules that depend on specific days, times, and sequences, such as "every 10 minutes on weekdays between 9 AM and 5 PM" or specific intervals within a set range.

For reference, the standard format is

```
┌──────────── Minute (0 - 59)
│ ┌──────────── Hour (0 - 23)
│ │ ┌──────────── Day of the month (1 - 31)
│ │ │ ┌──────────── Month (1 - 12)
│ │ │ │ ┌──────────── Day of the week (0 - 7) (Sunday = 0 or 7)
│ │ │ │ │
│ │ │ │ │
* * * * * CRON Expression (five parts only)
```

Here are some examples of complex and conditional CRON expressions:

CRON Expression	**Description**
0 12 5 * *	Every fifth day of the month at noon
15 10 15 * *	At 10:15 AM on the 15th of every month
0 0 9-17 * * 1-5	Every hour on the hour from 9:00 AM to 5:00 PM, Monday through Friday
*/7 * * * *	Every 7 minutes
5 * * * *	Every hour, at 5 minutes past the hour
*/10 14-15 * * 1-5	Every 10 minutes from 2:00 PM to 2:50 PM daily (Mon to Fri only)
*/10 14-15 * * *	Every 10 minutes from 2:00 PM to 2:50 PM daily (Mon to Sun)
15 8 15 6 *	At 8:15 AM on June 15th of every year
0 5 1/2 * *	Every other day at 5:00 AM

30 1 1 1 *	At 1:30 AM on the first day of every year
*/15 9-17 * * 1-5	Every 15 minutes during business hours (9:00 AM to 5:00 PM), Monday through Friday
0 8-18/2 * * *	Every 2 hours from 8:00 AM to 6:00 PM daily
0 8-18/3 * 1,3,6,9 1-5	Every 3 hours from 8:00 AM to 6:00 PM, Monday through Friday, only in specific months
*/7 6,18 * 2,4,6 1-5	Every 7 minutes at 6:00 AM and 6:00 PM on even months, Monday through Friday
0 9-17/4 * *	Every 4 hours from 9:00 AM to 5:00 PM only in January

CRON Examples with Exceptions

Standard CRON syntax itself does not directly support "exceptions" (like skipping certain dates), but you can work around this limitation by combining multiple CRON jobs or handling exceptions programmatically in scripts that are part of the activities invoked by the CRON schedules/process design.

For reference, the standard format is

```
┌──────────── Minute (0 - 59)
│ ┌──────────── Hour (0 - 23)
│ │ ┌──────────── Day of the month (1 - 31)
│ │ │ ┌──────────── Month (1 - 12)
│ │ │ │ ┌──────────── Day of the week (0 - 7) (Sunday = 0 or 7)
│ │ │ │ │
│ │ │ │ │
* * * * * CRON Expression (five parts only)
```

Here are some ways to handle exceptions or special cases:

CRON Expression	**Description**

0 9 * * 1-5	Weekdays Only, Except Specific Days (e.g., Skip Holidays) **Description**: Runs at 9:00 AM Monday through Friday. **Exception**: If you want to skip certain dates (like holidays), you could set up the process to check a list of holidays in your code and skip those dates programmatically.
0 0 * * 1-5	Every Day at Midnight, Except on Weekends **Description**: Runs every weekday at midnight, excluding Saturdays and Sundays.
0 0 1 * 1-5	First Day of the Month, Except Weekends **Description**: Runs at midnight on the first weekday of every month (if the 1st falls on a weekend, it will not run).
0 9-11,13-17 * * 1-5	Every Hour During Business Hours, Except Lunch Hour **Description**: Runs every hour on the hour, from 9:00 AM to 5:00 PM, excluding the lunch hour (12:00 PM), Monday through Friday.
0 0 0 * * 5	Every Friday, Except the Last Friday of the Month **Description**: Runs every Friday, but to exclude the last Friday, add a conditional check in your code to skip if it's the last Friday of the month.
*/15 0-1,3-23 * * *	Every Quarter Hour, Except from 2:00 AM to 3:00 AM **Description**: Runs every 15 minutes, except between 2:00 AM and 3:00 AM (useful for daylight saving time shifts or other maintenance times).
0 0 1-14,16-31 * *	Runs on Specific Days of the Month, Except the 15th. **Description**: This skips the 15th day of the month.
0 0 1 * *	Every Month on the 1st, Unless it Falls on a Weekend **Description**: This runs on the 1st of the month but include a script step in the process to check/skip if it falls on a Saturday or Sunday.

Running a Maintenance Process

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

Manually: To run a maintenance process manually

1. Click the icon next to its name on the Maintenance Processes page or click the **Run Now** button at the bottom of the Maintenance Process page. You can do this immediately after configuring a new process, or you can manually run a previously configured process:

    a. click **Report Warehouse**, then click **Maintenance Management** on the Navigation Tree and select **Maintenance Processes**.

    b. Click the name of the process you want to run.

    c. Click the **Run Now** button at the bottom of the Maintenance Process page.

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

Scheduled: To execute a maintenance process via a scheduled runtime

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

Editing a Maintenance Process

Maintenance Processes can be edited via the Maintenance Process page.

To edit a maintenance process

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

Deleting a Maintenance Process

If you no longer need a maintenance process, you can delete it.

To delete a maintenance process

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

Automator

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

Features and Functions

Automator has the following capabilities:

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

Server Status

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

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

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

- **Refresh Status**
    - **Interval (in seconds)**: Specify the refresh status interval in seconds.

- **Refresh Now**: Click the button to refresh status now.
- **Automatic**: Select this check box to refresh status at the specified interval automatically.

- **Server Status**
  - **Info from database**
    - **Standard Processes**
      - **Custodian Status**: Shows whether the custodian schedule is running.
      - **Jobs/Processes Currently Executing**: Shows the number of jobs in "Running" status. 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 all jobs with the following status: "Process ready to run," "Process awaiting retry after a failed run attempt," "Process ready to run," "Job running," "Export failed and will be retried," "Process resolving Web addresses in a manifest," "Dispatcher host is not responding," and “Process is running PreProcess script." 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 without completed dates and all other Visual Processes except those with the following status: "Process awaiting retry after a failed run attempt," "Orphan Job," and "Dispatcher host is not responding." 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 Visual Processes without completed dates. 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**
  - **Orphan Automator Jobs**: Shows the number of jobs that are stuck in the running state and belong to hosts that do not exist in the current configuration.
    - **Reset**: Clicking this button executes the orphan Automator jobs on existing hosts
    - **Cancel**: Clicking this button cancels the orphan Automator jobs and sets them in canceled state (**NOTE**: For orphan Automator jobs with schedules, they will be executed accordingly on the basis of the schedules specified and will not be placed in canceled state)
  - **Stuck Automator Jobs**: Shows the number of jobs that are stuck in the execution state because the host that performed these jobs is not responding.
    - **Reset**: Clicking this button executes the stuck Automator jobs on available hosts
    - **Cancel**: Clicking this button cancels the stuck Automator jobs and set in canceled state (**NOTE**: For stuck Automator jobs with schedules, they will be executed accordingly on the basis of the schedules specified and will not be placed in canceled state)
  - **Job Processing Status:** Shows the current job processing status.

- **Start:** Click this button starts the Automator Dispatcher Service Host. Starting the dispatcher enables the running of new jobs.
- **Stop:** Click this button stops the Automator Dispatcher Service Host. Stopping the dispatcher disables the 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 Monarch Data Pump products. These earlier versions of Altair 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)

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:

Symbol	Description
~	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 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 Standard Process

Once you have configured and saved a Standard 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 export performance.

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:
   - **Source Type**: Choose a **local** source and 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 of displayed in the **Named Path Content** box.
   - **Path (for Local Source Type)/File Path (for Named Paths)**: You may use the naming macros to specify the file path. Click the Insert Macro icon and select the naming macros from the drop-down list.
   - 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.

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

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 Standard 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 a location for processes exported with defined locations.

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.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**, or **Cron**.
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 please 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 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.

- **Cron**: Select this option if you want to set a cron schedule.
  - **Expression**: Enter the cron expression and the press the **TAB** key to validate it. The following guidelines are available for cron expressions:
    - The maximum length is 64 characters.
    - The expression should have 5 fields only, as shown in the following format: **<minutes> <hours> <day of the month> <month> <day of the week>** (e.g., 20 20 ? * ?).
    - The "seconds" field is not accepted; the first cron field specified is interpreted as "minutes."
    - For the "month" field, the numeric values 1 to 12 are accepted; abbreviations such as JAN and DEC are not allowed.
    - For the "day of the week" field, the numeric values 0 to 6 are accepted; abbreviations such as SUN and SAT are not allowed.

NOTE: Detailed documentation for cron, including descriptions and examples, is available here.

- In the **At** check box set the time when the current process will be launched following the **HH:MM** format.

CRON Functionality

This section describes in detail the CRON functionality in Monarch Server. (For French, click here.)

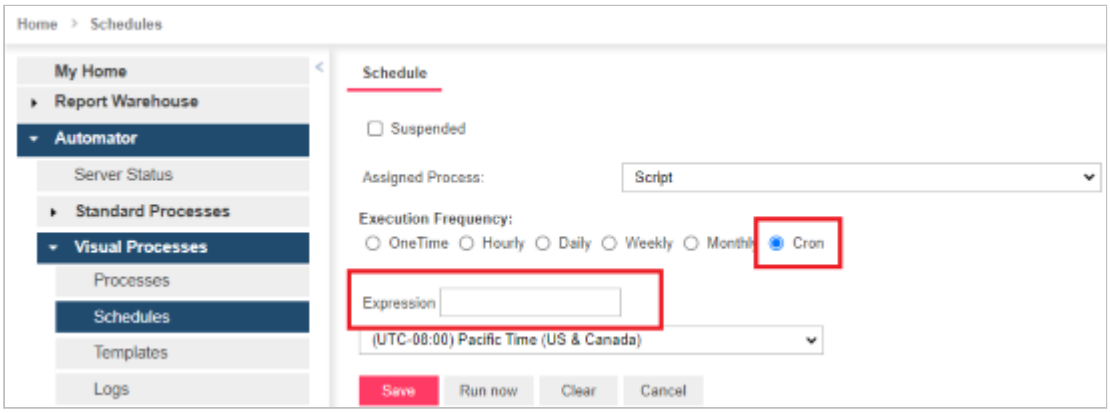
What is CRON?

CRON is a time-based job scheduler that is traditionally used in Unix-like operating systems but has since been inherited by others. It is designed to automatically execute commands or scripts at specific times or intervals. It is commonly used to automate repetitive tasks, such as backups, email notifications, system maintenance, and data processing.

CRON in Monarch Server

Monarch Server has background modules that run continuously and execute scheduled tasks.
Once configured, the modules check for associated task schedules to run as frequently as every minute (assuming correct instance configuration) based on internally stored CRON expressions, which specify the exact times or intervals when tasks should run.

In the Monarch Server scheduling interface (version 2024.1+), a new option allows for the provision of a user-defined CRON string as opposed to using the scheduling interface to define the same via the options presented by (and limited to) the interface options.



When the CRON option is selected, you can either enter or paste the CRON string desired in the Expression box.

Monarch Server CRON Expression Format

CRON expressions consist of five fields that define the schedule, plus the command or script to execute. The standard format is as follows:

```
┌──────────── Minute (0 - 59)
│ ┌──────────── Hour (0 - 23)
│ │ ┌──────────── Day of the month (1 - 31)
│ │ │ ┌──────────── Month (1 - 12)
│ │ │ │ ┌──────────── Day of the week (0 - 7) (Sunday = 0 or 7)
│ │ │ │ │
│ │ │ │ │
* * * * * CRON Expression (five parts only)
```

Monarch Server CRON Expression Symbols

Symbol	Description
*	Wildcard, meaning "every" value in that field
,	Allows specifying multiple values in the same field (e.g., 1,15 for the minute field means 1st and 15th minute)
-	Defines a range of values (e.g., 9-17 for the hours field means from 9 AM to 5 PM)
/	Defines a step value (e.g., */15 for the minutes field means every 15 minutes)

NOTE:

Not all CRON systems support enhanced characters such as **L**, **W**, **#**, or **?**.

Standard CRON (per the Monarch Server implementation) supports basic characters such as *****, **,**, **-**, and **/**.

Advanced characters such as **L**, **W**, **?**, and **#** are typically found in Quartz or other enhanced schedulers and are NOT supported in Monarch Server.

Examples of CRON Expressions

Here are some typical CRON jobs with their schedules:

CRON Expression	Schedule
* * * * *	Every minute
0 0 * * *	Every day at midnight
0 0 * * 1	Every Monday at midnight
*/15 9-17 * * 1-5	Every 15 minutes, 9 AM to 5 PM, Monday through Friday

Monarch Server has historically stored user-defined schedules internally using the CRON syntax. However, post-Monarch Server 2024.1, administrators can define schedules using CRON syntax explicitly, thus allowing for more efficient and complicated and customised scheduling definitions.

Generating CRON Expressions

CRON expressions can be created using the following options:

- Copying/pasting of existing strings (assuming they comply with the traditional five-part CRON/CronTab notation used in Monarch Server)
- Deriving expressions directly based on existing CRON knowledge and experience
- Using the samples provided in this document (with potential modification as desired)
- Using online tools and resources that provide other examples or assist in creating new expressions interactively. Some useful examples of these resources are as follows:
    - Crontab.guru - The cron schedule expression generator: a simple CRON expression builder/editor;
    - Cron Expression Generator | Uptimia: allows you to provide a natural language-based requirement which the site will attempt to parse into a CRON string;
    - Cron job editor: multiple cron jobs, calendar view, AWS & Vercel cron support | CronTool: simple CRON editor/debugger with an on-screen calendar to help visualize the subsequent execution event(s); a more advanced version of CronTool allowing for multiple CRON expressions to be “stacked” to see their combined execution events.

NOTE: Some of the CRON schedules will require script to be added as part of the process to enable exclusions and exceptions outside of the scope of the CRON statement or construct limitations.

Benefits of CRON Expressions

Using CRON strings for scheduling offers several advantages.

1. **Precision and Flexibility**
    - **Fine-Grained Scheduling**: CRON strings allow for precise scheduling down to the minute. You can specify tasks to run at exact times (e.g., every hour, specific days, or monthly).
    - **Complex Schedules**: CRON strings can handle both simple and complex scheduling needs, such as "every 15 minutes during business hours." This flexibility is essential for tasks that require specific timing.
    - **Patterns and Recurrence**: With symbols like * (wildcard), / (step), and ranges (-), CRON can define recurring intervals without having to specify each individual time.
2. **Consistency and Reliability**
    - **Automated Execution**: Once a CRON job is set up, it can run automatically at the scheduled times without any manual intervention. This reliability ensures that critical tasks can be consistently performed on schedule, assuming a correctly configured Monarch Server instance.
    - **Repeatability**: CRON jobs run consistently across time zones, system reboots, and other environmental changes, making them highly dependable for tasks that must occur regularly.
3. **Simplicity and Ease of Use**
    - **Concise Syntax**: CRON strings use a compact, standardized syntax that’s easy to understand and customize once learned. A comprehensive list is included later in this document.
    - **Minimal Configuration**: CRON strings require minimal setup and can be easily configured by an administrator in the Monarch Server Admin UI. This simplicity is particularly helpful for system administrators managing multiple tasks or where a task requires multiple schedules.
4. **Versatile Applications Across Systems**
    - **Wide Compatibility**: CRON is available on almost all Unix-like systems, making it a universal scheduling tool across Linux, macOS, and other systems. This portability is especially advantageous in heterogeneous environments.
    - **Suitable for Various Tasks**: CRON can schedule almost any command or script, including a maintenance script, a data backup, or an email notification, making it versatile for many automation needs.
5. **Customizability for Complex and Conditional Scheduling**
    - **Exceptions and Conditional Logic**: By combining CRON with process scripting or checks in the executable code, you can create conditional schedules (e.g., excluding certain dates and adjusting for holidays).

Common CRON Examples

Below is a list of common CRON expressions with descriptions of what they do. These examples can help cover most typical scheduling needs.

For reference, the standard format is

```
┌──────────── Minute (0 - 59)
│ ┌──────────── Hour (0 - 23)
│ │ ┌──────────── Day of the month (1 - 31)
│ │ │ ┌──────────── Month (1 - 12)
│ │ │ │ ┌──────────── Day of the week (0 - 7) (Sunday = 0 or 7)
│ │ │ │ │
│ │ │ │ │
* * * * * CRON Expression (five parts only)
```

CRON Expression	Description
* * * * *	Every minute
*/5 * * * *	Every 5 minutes
0 * * * *	Every hour, on the hour
0 */2 * * *	Every 2 hours, on the hour
0 0 * * *	Every day at midnight
0 9 * * *	Every day at 9:00 AM
0 0 * * 1	Every Monday at midnight
0 0 1 * *	The first day of every month, at midnight
0 0 1 1 *	Every January 1st at midnight

0 0 * * 0	Every Sunday at midnight
*/15 9-17 * * 1-5	Every 15 minutes during business hours (9:00 AM to 5:00 PM) Monday through Friday
0 12 * * 1-5	Every weekday at noon
0 0 1-7 * 0	The first Sunday of every month, at midnight
0 0 */3 * *	Every third day
*/10 0 1 1 *	Every 10 minutes on January 1st, between midnight and 12:59 AM
0 0 25 12 *	Every Christmas Day at midnight
30 14 1 * *	The first day of every month at 2:30 PM
0 8,16 * * *	Every day at 8:00 AM and 4:00 PM
*/5 8-10 * * *	Every 5 minutes, between 8:00 AM and 10:59 AM
0 0 1-7 * 0	The first Sunday of every month
0 6,18 * * *	Every day at 6:00 AM and 6:00 PM
*/30 * * * *	Every 30 minutes
0 22 * * 1-5	Every weekday at 10:00 PM
0 7 * * 1	Every Monday at 7:00 AM
*/2 * * * 3	Every 2 minutes, on Wednesdays
15 14 1 * *	Every month on the 1st, at 2:15 PM
0-4 14 * * *	Every minute starting at 2:00 PM and ending at 2:05 PM daily
5 * * * *	At 5 minutes past every hour
15 10 15 * *	At 10:15 AM on the 15th day of every month

15 10 15 * 1-5	At 10:15 AM every Monday through Friday
0 0 15-21 * 5	At 10:15 AM on the third Friday of every month

CRON Examples with Conditions

Conditional CRON expressions are powerful tools for defining schedules that depend on specific days, times, and sequences, such as "every 10 minutes on weekdays between 9 AM and 5 PM" or specific intervals within a set range.

For reference, the standard format is

```
┌──────────── Minute (0 - 59)
│ ┌──────────── Hour (0 - 23)
│ │ ┌──────────── Day of the month (1 - 31)
│ │ │ ┌──────────── Month (1 - 12)
│ │ │ │ ┌──────────── Day of the week (0 - 7) (Sunday = 0 or 7)
│ │ │ │ │
│ │ │ │ │
* * * * * CRON Expression (five parts only)
```

Here are some examples of complex and conditional CRON expressions:

CRON Expression	**Description**
0 12 5 * *	Every fifth day of the month at noon
15 10 15 * *	At 10:15 AM on the 15th of every month
0 0 9-17 * * 1-5	Every hour on the hour from 9:00 AM to 5:00 PM, Monday through Friday
*/7 * * * *	Every 7 minutes
5 * * * *	Every hour, at 5 minutes past the hour
*/10 14-15 * * 1-5	Every 10 minutes from 2:00 PM to 2:50 PM daily (Mon to Fri only)

*/10 14-15 * * *	Every 10 minutes from 2:00 PM to 2:50 PM daily (Mon to Sun)
15 8 15 6 *	At 8:15 AM on June 15th of every year
0 5 1/2 * *	Every other day at 5:00 AM
30 1 1 1 *	At 1:30 AM on the first day of every year
*/15 9-17 * * 1-5	Every 15 minutes during business hours (9:00 AM to 5:00 PM), Monday through Friday
0 8-18/2 * * *	Every 2 hours from 8:00 AM to 6:00 PM daily
0 8-18/3 * 1,3,6,9 1-5	Every 3 hours from 8:00 AM to 6:00 PM, Monday through Friday, only in specific months
*/7 6,18 * 2,4,6 1-5	Every 7 minutes at 6:00 AM and 6:00 PM on even months, Monday through Friday
0 9-17/4 * *	Every 4 hours from 9:00 AM to 5:00 PM only in January

CRON Examples with Exceptions

Standard CRON syntax itself does not directly support "exceptions" (like skipping certain dates), but you can work around this limitation by combining multiple CRON jobs or handling exceptions programmatically in scripts that are part of the activities invoked by the CRON schedules/process design.

For reference, the standard format is

```
┌──────────── Minute (0 - 59)
│ ┌──────────── Hour (0 - 23)
│ │ ┌──────────── Day of the month (1 - 31)
│ │ │ ┌──────────── Month (1 - 12)
│ │ │ │ ┌──────────── Day of the week (0 - 7) (Sunday = 0 or 7)
│ │ │ │ │
│ │ │ │ │
* * * * * CRON Expression (five parts only)
```

Here are some ways to handle exceptions or special cases:

CRON Expression	Description
0 9 * * 1-5	Weekdays Only, Except Specific Days (e.g., Skip Holidays) **Description**: Runs at 9:00 AM Monday through Friday. **Exception**: If you want to skip certain dates (like holidays), you could set up the process to check a list of holidays in your code and skip those dates programmatically.
0 0 * * 1-5	Every Day at Midnight, Except on Weekends **Description**: Runs every weekday at midnight, excluding Saturdays and Sundays.
0 0 1 * 1-5	First Day of the Month, Except Weekends **Description**: Runs at midnight on the first weekday of every month (if the 1st falls on a weekend, it will not run).
0 9-11,13-17 * * 1-5	Every Hour During Business Hours, Except Lunch Hour **Description**: Runs every hour on the hour, from 9:00 AM to 5:00 PM, excluding the lunch hour (12:00 PM), Monday through Friday.
0 0 0 * * 5	Every Friday, Except the Last Friday of the Month **Description**: Runs every Friday, but to exclude the last Friday, add a conditional check in your code to skip if it’s the last Friday of the month.
*/15 0-1,3-23 * * *	Every Quarter Hour, Except from 2:00 AM to 3:00 AM **Description**: Runs every 15 minutes, except between 2:00 AM and 3:00 AM (useful for daylight saving time shifts or other maintenance times).
0 0 1-14,16-31 * *	Runs on Specific Days of the Month, Except the 15th. **Description**: This skips the 15th day of the month.
0 0 1 * *	Every Month on the 1st, Unless it Falls on a Weekend **Description**: This runs on the 1st of the month but include a script step in the process to check/skip if it falls on a Saturday or Sunday.

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
- The status of the job
    - Registered
    - Process resolving Web addresses in a manifest
    - Process is running PreProcess script
    - Process ready to run
    - Process awaiting entry after a failed run attempt
    - Process failed and not scheduled for retry
    - Process canceled before jobs were launched
    - Job ready to be dispatched
    - Job deferred while a parallel job was running
    - Job running
    - Export failed and will be retried
    - Job completed without errors
    - Job failed
    - Job canceled before completion
    - Dispatcher host is not responding (a job has been executed on a host that is currently not responding)
    - Orphan job (a job has been executed on a host that does not exist in the current configuration)
- 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
- SDK verison

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:

Symbol	Description
~	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. Specifically, jobs deleted from the UI are subsequently deleted from the database. 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 **Jobs** page do one of the following:
   - Select the job you want to delete, by clicking anywhere on the row, and then click , or
   - Click at the end of the row with the job you want to delete. A dialog box appears.

NOTE: Multiple jobs selection is possible.

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

Distributions

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

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

There are two types of distributions that can be specified:

- **Project Distributions**: These distributions apply only to the input, export and portable report files of the specified project.

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

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

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

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

Adding a Distribution

To create a Global Distribution

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

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

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

Editing a Distribution

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

To edit a distribution

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

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

For details about how to change distribution parameters please refer to Adding a Distribution.

Adding a Distribution Action

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

To add an action for a distribution

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

Translate Distribution Action

To add a translate action

1. Select the **Translate** option on the Action page.

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

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

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

Move Distribution Action

To add a move action

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

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

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

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

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

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

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

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

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

RSS Distribution Action

To add an RSS action

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

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

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

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

7. Click the **Channel** title to provide the channel parameters:
    - **Title**: Enter channel title.
    - **Channel Description**: Provide channel description.
    - **Link**: Enter the name of link to the file. (**NOTE**: If you leave the **Title**, **Channel Description**, and **Link** fields blank the default values will be automatically assigned.)
    - **Max Items**: Specify the maximum number of items.
    - **Unlimited**: Select this check box if you want to set the unlimited number of items.
8. Click **Save** to save the action.

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

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

Log Distribution Action

To add a log action

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

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

Copy Distribution Action

To add a copy action

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

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

   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.

NOTE: 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

To add an 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.

- **Summaries: 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's 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

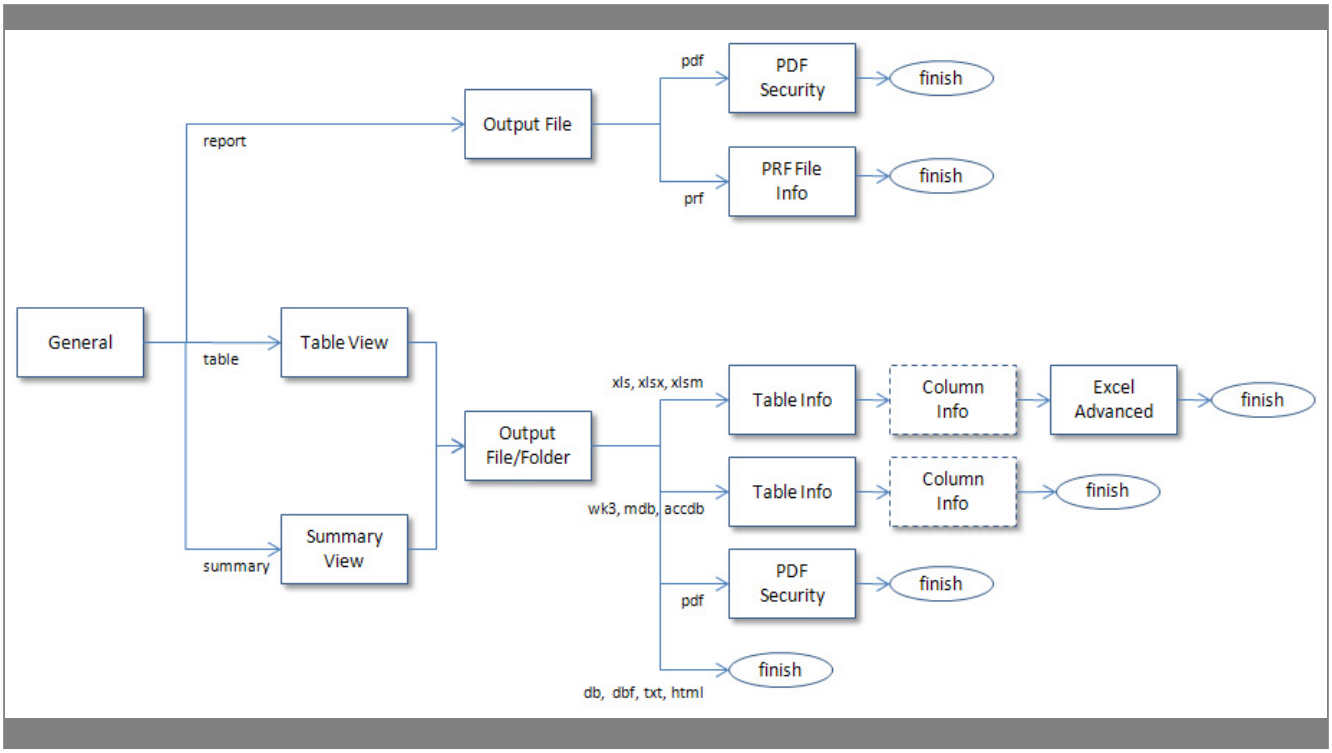
- PDF Security screen
- PRF Info screen
- Tableau Server screen
- OleDb Info screen
- Table Info screen
- Column Info screen
- Select Columns to Map dialog
- Excel Advanced Features screen

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 you specify **.pdf** as the file extension in the **Output File** screen of the **Export Wizard** and then click **Next**. 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

This screen displays when you specify **.prf** as the file extension in the **Output File** screen of the **Export Wizard** and then click **Next**. Use this dialog to specify settings for a portable report.

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 you specify **.hyper** as the file extension in the **Output File** screen of the **Export Wizard** and then click **Next**. Use this screen to specify the settings for publishing to Tableau Server.

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

- **Publish to Tableau Server**: Select this option to display the 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**: Select this option to use a password file in .txt format.
- **Password**: The password to the Tableau Server (**NOTE**: If the **Use Password File** option is selected, you need 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 source with the data from the export table (**NOTE**: Date modified is changed);
    - **Replace data**: Replaces the data in the data source with the 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)
- 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

NOTES:

- 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.
- In Visual Process Designer, you can set multiple instances of a visual process to run in parallel by selecting the **Allow multiple instances run in parallel** checkbox in the Process Global Settings. If this option is not selected, running a Visual Process in the Processes page while another instance is already running will result in the following error message: *[Process name]: Another instance of process is still running*.

From the Processes page, you may also

- Import Processes
- Export Processes
- Migrate Standard Processes

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:

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

Adding a Visual 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 Visual 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 Visual 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, 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 the 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** • **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**, or **Cron**.
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.
3. 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.

You may also use the naming macros to specify the file path (for local and named paths).

Click the Insert Macro icon and select the naming macros from the drop-down list.

4. 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.
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** 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.) You may also use the naming macros to specify the file path (for local and named paths). Click the Insert Macro icon and select the naming macros from the drop-down list. 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, 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. 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. **NOTE**: When a process is being run in Designer mode, the **Save and Run** button is enabled. If the **Allow multiple instances run in parallel** checkbox in Process Global Settings is selected, another instance of the process can be run. If the option is not selected, clicking **Save and Run** again while the process is running will result in the following error: *[Process name]: Another instance of process is still running*. In Log mode, the **Save and Run** button is disabled while a process is being run.
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.
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** button to actualize your changes by running a process with updated settings. Click **Stop** to cancel the execution of the process. **NOTE**: When a process is being run, the **Save and Run** button is disabled regardless of the selection in Process Global Settings. **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 (XML)** button to export the log to an XML file. Click the **Export Log (HTML)** button to export the log to an HTML file. Click the **Designer Mode** icon to leave the Log Mode view.
Process Global Settings	Click to modify the workspace validation and process settings of the Visual Process in the **Process Global Settings** window. • Select the **Allow multiple instances run in parallel** checkbox to enable multiple instances of a process to be run. This checkbox is selected by default; if it is not selected, you will not be able to run more than one instance at a time, as indicated in the following error message: *[Process name]: Another instance of process is still running*. • 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	Click 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: ◦ 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 (reports in .txt, .prn, and .pdf formats) located in the web • HTTP • Secure HTTP • FTP • Secure FTP • Amazon S3 • Azure Data Lake • Azure Blob Storage • Oracle Cloud • Google Cloud	• **Name** — name of the graphic element. • **Input Path** — 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**: Click the **Browse** button to open the **Browse for File** window from which you can view system containers and select your web input file. If some web input file requires authentication, you should add the Credentials element and connect your **Web Input** element to it. In the properties of the **Credentials** element, you should specify the necessary parameters. For web inputs, the replacement of original file sources is only applicable to report-dependent load plans. For web-based database inputs (e.g., .csv and .xlsx), use the Database Input element.
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. **NOTES**:

- In addition to Excel file types .xls and .xlsx, .xlsb and .xlsm file types may be used as database file input.
- For local or web files, *local*, *named paths*, and *web* source types are available.
- For web source types, you can set the address to FTP/SFTP, HTTP location, HTTPS, Amazon S3, Azure Data Lake, Azure Blob Storage, Oracle Cloud, and Google Cloud.
- For the web-based systems (i.e., Amazon S3, Azure Data Lake, Azure Blob Storage, Oracle Cloud, and Google Cloud), click the **Browse** button to open the **Browse for File** window from which you can view the system containers and select your input file.

- 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.
- To apply filtering to the table during the export, select the **Apply Filter** check box and click the **Change** button to specify necessary filters.
- 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.

		• 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**. 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.

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.

NOTES:

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. This information can be viewed by clicking the information icon below the **Model Path** field.

The table below specifies the functionality of each **Model** element.

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. When executing processes with external lookup, the following error message is displayed in the logs when the external lookup data source in the model is missing or invalid: An external lookup data source 'path_to_the_external_lookup' is either missing or invalid.

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. **NOTE:** To view the information of the selected workspace, including its table inputs, exports, filters, sorts, summaries, and other settings, click the information icon below the **Workspace Path** field. Actions to select the workspace path: 1. Click the **Browse** button to open the Browse dialog. 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**.

NOTE: The following alert displays when you change the workspace path of an existing workspace element: *Load Plan should be reselected and reapplied in DataPrep Export/Verify Workspace settings because another workspace was specified*. (It is impractical to assume that the Load Plan is still valid, given the Workspace has changed, thus, requiring this step to confirm/refresh accordingly).

- **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

NOTE: Workspace validation is sensitive to data volume and connection speed and is thus disabled by default. For web/cloud-based load plans, you may select the **Disable validation optimization** checkbox to disable the automatic downloading/parsing of files from the web.

- **Load Plans** — the list of available export load plans, including:
  - **Provider type**. This shows the type of 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.

NOTES:

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.

Database Input is required for Excel formats and must include the extension even with wildcards.

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 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, modify the following settings: Key File Path (.json format), Property ID, Categories, Dimensions, Metrics, and Selected Columns. 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.

		NOTE: When the load plan comes with a change list, this change list is labeled as “Default.” All new change lists added to the workspace’s change history are labeled according to their names in the Server Library. When starting a process via manifest, the existing change list in the process can be overwritten with a change list from the Server Library.
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. • **Import Export from Workspace** — allows you to import an export previously saved with the connected workspace. Click this button to open the **Import Export Definition** dialog box. In the **Export Name** dropdown menu, select the workspace export you want to import. The corresponding details of the export (e.g., load plan, export type) are displayed. Click OK to proceed. The settings of the selected export are then applied to the DataPrep Export item settings. **NOTE**: The advanced settings for a predefined Excel export from the selected export definition will also be fetched and applied to the DataPrep Export item settings. A label of **Excel Advanced Options: *Default*** indicates that the selected export definition to be imported has no advanced Excel export settings. A label of **Excel Advanced Options: *Custom*** indicates that the selected export definition has at least one advanced Excel export setting. However, if the **Enable Advanced Excel features** for existing exports of the current workspace is disabled, the advanced Excel export settings of the selected export definition are not applied to the DataPrep export item at all. Note that the same does not apply to new exports created in Monarch Server. • **Export File System** — the file system to which your table will be exported. This setting is available for the following export types: ◦ CSV ◦ Microsoft Excel

- Microsoft Access
- Qlik
- Tableau Hyper
- SAS Transport Format
- SAS V8 Transport Format
- JSON
- XML

The following file systems are available:

- Local or mounted disk
- Amazon S3
- Azure Data Lake Storage Gen2
- Azure Blob Storage
- Oracle Cloud Infrastructure
- Google Cloud Storage

- **Export Type** — the type of export file. Further settings depend on the type of file and are described as advanced options below.
- **Export Destination** — path to required export file. Click the Browse button to modify the destination path for the *local or mounted disk* file system or to open the **Browse For File** window from which you can view the containers of your selected export file system and set the file to be used as the export destination. Then, click **Ok** to proceed.

NOTES:

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.

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.

- **Export Options** — allows you to specify the filter, summary, and sort settings for the export

Filtering:

- **Select filter**: From the dropdown list, choose the filter you want to apply to the export. When **All filters** is selected, you will be asked to specify the **Automatic Naming** setting:
  - **By Files**: produces a file for each filter
  - **By Tables**: produces a table for each filter

Select a summary:

> **NOTE**: This option is available for the following file types: CSV, Microsoft Excel, Microsoft Access, XML, JSON, Tableau, SAS Transport Format, and Qlik.

- **Summary**: From the dropdown list, choose the summary to apply to the export. Selecting **All summaries** will let you specify the **Automatic Naming** setting:
  - **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.
- **Measure**: Allows you to specify the measure to apply to the summary selected. This is activated when the summary has a defined column key. From the dropdown list, you can also choose **All, by key** to include all measures grouped by key or **All, by measure** to include all measures grouped by measure.
- **Split**: Check **Each distinct value of the first key makes a separate file or table** to split the summary export according to each value change in the first key (i.e., the leftmost one). The **Automatic Naming** setting can then be modified:
  - **By Files**: produces a file for each distinct first key
  - **By Tables**: produces a table for each distinct first
- **Drill level**: Set the drill level to apply to the summary being exported. 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.

Sorting

		◦ **Select Order**: From the dropdown list, choose the sort to apply to the export. **NOTE**: When the selected table has a row limit applied, you can proceed with the export by exporting either all rows or limited rows. Specifically, the following options are available: ▪ **Export all rows** : the export file will contain all the rows from the workspace ▪ **Export limited rows (Limit: X number of rows)**: the export file will contain only a limited number of rows, as set in Data Prep Studio. The option to export either all rows or limited rows is only available when one load plan is selected. It cannot be modified when multiple load plans are selected for export.

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 checkbox if you want to parse the inputs for specific sheets, and then click **Apply to…**
    - When selected, changes can be applied to the following:
        - All worksheets
        - First sheet
        - Last sheet
        - Selected Sheet by Name
        - Selected Sheet by Number
    - 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 by clicking **Apply to…**:
    - All worksheets
    - First sheet
    - Last sheet

- Selected Sheet by Name
- Selected Sheet by Number

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.

   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:

     - **Equal to**: Applies to numeric data, dates, and text; returns all values that contain the specified filter value
     - **Not Equal To**: Applies to numeric data, dates, and text; returns all values except those that contain the specified filter value
     - **Greater Than**: Applies to numeric data and dates; returns all values that are greater than the specified filter value
     - **Greater Than Equal**: Applies to numeric data and dates; returns all values that are greater than or equal to the specified value
     - **Less Than**: Applies to numeric data and dates; returns all values that are less than the specified filter value
     - **Less Than Equal**: 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
     - **Starts With**: Applies to text; returns all values that start with the specified filter value
     - **Ends With**: Applies to text; returns all values that end with the specified filter value

Advanced options for DataPrep Export element

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,
    - select **Add data** to add the new data to the existing one,
    - select **Skip** to stop the export operation from overwriting existing data.
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** checkbox to include headers in your file.

7. Select the **Append DOS end-of-life character (x1A)** checkbox to add a DOS EOF character to the export file.
8. Select the **Legacy compatibility mode – Ignore format customizations during export** checkbox if you want to use the Monarch Classic style formatting instead of the column format customizations during the export of numeric and date columns.

> **NOTES**
>
> When this option is selected, numeric values are exported using the current system's localized decimal separator. For date values, the following options are available:
>
> - yyyy-mm-dd
> - yyyymmdd
> - Use current Windows Regional Settings (based on the settings of the server where the host is installed)
>
> For Monarch Server, this would typically be based on the Service Account for the Monarch Server Agent Service localization specifically (or, if running Monarch Server in console mode, the User Account context in which the console is running).

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,
   - select **Add data** to add the new data to the existing one,
   - select **Skip** to stop the export operation from overwriting existing data.
2. Click **Column sizes** to adjust the widths of the columns in your file.

Additional options for Altair Panopticon file types

1. In the **When output file exists** dropdown 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 existing data.
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. When **Add data** is selected in the **When output file exists** dropdown menu, the **When table exists** dropdown menu becomes available with the following options:
   - select **Overwrite** to substitute the data table,
   - select **Append** to append records to the existing table,

- select **Update** to update existing rows,
- select **Update and append** to update existing rows and append new ones,
- select **Skip** to stop the export operation from overwriting existing data.

4. Select the **Suppress column header row** checkbox if you prefer to hide the column headers in the exported file.
5. Click the **Advanced Excel Export Options** button to set the parameters for the Excel export. The following settings are available:
    - **Security**
        - **Digital Signature**: Select this checkbox to apply a digital signature to the exported Excel file. By adding a digital signature, you establish the authenticity, integrity, and non repudiation of the export output. Select or change the certificate to be used by clicking the Certificate button. In the Select Certificate dialog box, specify the certificate source by choosing between *Store* and *File* in the dropdown menu.
            - *Store*: A list of the certificates currently available on your Certificate Store is displayed. Select a certificate from the list and click **OK**.
            - *File*: Navigate to the folder containing your certificate file and then select it. Enter the certificate password in the field provided. Click the **Get Certificate Information** button to display the details of the certificate.
    - **Filter**
        - **Add AutoFilter drop down lists to column labels in the Table**: Select this checkbox to enable Excel's AutoFilter feature so that it is ready and active upon opening the exported Excel file.
    - **Pivot**
        - **Include Pivot Table sheet**: Select this checkbox to modify the pivot settings. Click the **Pivot Table** button to display the **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.
6. Click the **Page Setup** button to specify the header/footer settings and other page settings for the Excel export in the **Excel Page Setup** window. Select the **Use workspace settings** checkbox to apply the default page settings of the workspace used. To make your own specifications, the following settings are available:
    - **Preferences**
        - **Page Orientation**: Set the desired page orientation (i.e., portrait or landscape).
        - **Paper Size**: Set the desired paper size (e.g., letter, legal, or executive).
        - **Page Margins**:
            - **Top**: Accepts a value representing the top margin. The default value is 0.5 inches.
            - **Left**: Accepts a value representing the left margin. The default value is 0.5 inches.

        - **Bottom**: Accepts a value representing the bottom margin. The default value is 0.5 inches.
        - **Right**: Accepts a value representing the right margin. The default value is 0.5 inches.
    - **Options**
        - **Show Column Titles**: Select this checkbox to display the column titles at the top of each page.
        - **Show Grid Lines**: Select this checkbox to display the grid lines between rows and columns.
  - **Header Layout/Footer Layout**
    - **Page Header display box**: This shows a page header of up to 3 lines, with each line displaying left-aligned, centered, or right-aligned text. Input text (e.g., workspace name) into the boxes accordingly.
    - **Use custom font**: Select this checkbox to modify the font settings, including the font style, font size, text color, and text style (i.e., bold, italic, underline). A sample of the text with the font settings applied is displayed on the right panel of the Excel Page Setup window.

7. Select the **Export All** checkbox if you want to export all columns; otherwise, click **Column Mapping** to map the export columns to the destination columns.

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,
    - select **Add data** to add the new data to the existing one,
    - select **Skip** to stop the export operation from overwriting existing data.
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. When **Add data** is selected in the **When output file exists** dropdown menu, the **When table exists** dropdown menu becomes available with the following options:
    - select **Overwrite** to substitute the data table,
    - select **Append** to append records to the existing table,
    - select **Skip** to stop the export operation from overwriting existing data.
4. Select the **Suppress column header row** checkbox if you prefer to hide the column headers in the exported Excel file.
5. When **Append** is selected from the **When table exists** dropdown menu, select the **Do not match column names** checkbox if you want the columns from potentially different tables to be left aligned regardless of column names or data types.
6. Click the **Advanced Excel Export Options** button to apply the following settings:

   <u>For Table Exports:</u>

- **Digital Signature**: Select this checkbox to apply a digital signature to the exported Excel file. By adding a digital signature, you establish the authenticity, integrity, and non repudiation of the export output. Select or change the certificate to be used by clicking the Certificate button. In the Select Certificate dialog box, specify the certificate source by choosing between *Store* and *File* in the dropdown menu.

  - *Store*: A list of the certificates currently available on your Certificate Store is displayed. Select a certificate from the list and click **OK**.

  - *File*: Navigate to the folder containing your certificate file and then select it. Enter the certificate password in the field provided. Click the **Get Certificate Information** button to display the details of the certificate.

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

- **Include Pivot Table sheet**: Select this checkbox to modify the pivot settings. Click the **Pivot Table** button to display the **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.

For Summary Exports:

- **Digital Signature:** Select this checkbox to apply a digital signature to the exported Excel file. By adding a digital signature, you establish the authenticity, integrity, and non repudiation of the export output. Select or change the certificate to be used by clicking the **Certificate** button. In the **Select Certificate** dialog box, specify the certificate source by choosing between *Store* and *File* in the dropdown menu.

  - *Store*: A list of the certificates currently available on your Certificate Store is displayed. Select a certificate from the list and click **OK**.

  - *File*: Navigate to the folder containing your certificate file and then select it. Enter the certificate password in the field provided. Click the **Get Certificate Information** button to display the details of the certificate.

- **Embed the page setup header in the worksheet title**: Select this checkbox 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**: Select this checkbox 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**: Select this checkbox if you want Automator to attempt to translate subtotal and grand total calculations into Excel formulas in the exported Excel file.

- **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.

7. Click the **Page Setup** button to specify the header/footer settings and other page settings for the Excel export in the **Excel Page Setup** window. Select the **Use workspace settings** checkbox to apply the default page settings of the workspace used. To make your own specifications, the following settings are available:
    - **Preferences**
        - **Page Orientation**: Set the desired page orientation (i.e., portrait or landscape).
        - **Paper Size**: Set the desired paper size (e.g., letter, legal, or executive).
        - **Page Margins**:
            - **Top**: Accepts a value representing the top margin. The default value is 0.5 inches.
            - **Left**: Accepts a value representing the left margin. The default value is 0.5 inches.
            - **Bottom**: Accepts a value representing the bottom margin. The default value is 0.5 inches.
            - **Right**: Accepts a value representing the right margin. The default value is 0.5 inches.
        - **Options**
            - **Show Column Titles**: Select this checkbox to display the column titles at the top of each page.
            - **Show Grid Lines**: Select this checkbox to display the grid lines between rows and columns.
    - **Header Layout/Footer Layout**
        - **Page Header display box**: This shows a page header of up to 3 lines, with each line displaying left-aligned, centered, or right-aligned text. Input text (e.g., workspace name) into the boxes accordingly.
        - **Use custom font**: Select this checkbox to modify the font settings, including the font style, font size, text color, and text style (i.e., bold, italic, underline). A sample of the text with the font settings applied is displayed on the right panel of the Excel Page Setup window.
8. Select the **Export All** checkbox if you want to export all columns; otherwise, click **Column Mapping** to map the export columns to the destination columns.

Additional options for Microsoft Access file types

1. In the **When output file exists** dropdown 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. When **Add data** is selected in the **When output file exists** dropdown menu, the **When table exists** dropdown menu becomes available with the following options:
    - select **Overwrite** to substitute the data table,
    - select **Append** to append records to the existing table,
    - select **Update** to update existing rows,
    - select **Update and append** to update existing rows and append new ones,

    - select **Skip** to stop the export operation from overwriting existing data.

4. Select the **Export All** checkbox if you want to export all columns; otherwise, click **Column Mapping** to map the export columns to the destination columns.

5. When **Update** or **Update and append** is selected from the **When table exists** dropdown menu,modify the following:

    - **When update match not found**: elect 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.

Additional options for OLE DB and ODBC file types

1. Type in the **Connection String** or select it via the **Data Link Wizard** or **Connection Definitions** with all the required data.

2. Select the **Include password** checkbox to input a password.

3. Select the **Bulk Behavior** checkbox to enhance export performance.

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

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

    - The **Use Transaction** checkbox 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 **Append** to append records to the existing table,

    - select **Update** to update existing rows,

    - select **Update and append** to update existing rows and append new ones.

    - select **Skip** to stop the export operation from overwriting existing data.

7. Select the **Export All** checkbox if you want to export all columns; otherwise, click **Column Mapping** to map the export columns to the destination columns.

8. When **Update** or **Update and append** is selected from the **When table exists** dropdown menu, modify the following:

    - **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.

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,
    - select **Add data** to add the new data to the existing one,
    - select **Skip** to stop the export operation from overwriting an existing data.

Additional options for Tableau Hyper file types

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

> **NOTE**: Export to Tableau is performed in .hyper 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**: Select this option to use a password file in .txt format.
    - **Password**: The password to the Tableau Server (**NOTE**: If the **Use Password File** option is selected, you need 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 source with the data from the export table (**NOTE**: Date modified is changed);
        - **Replace Data**: Replaces the data in the data source with the 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.

Additional options for IBM Cognos Analytics file types

1. Enter your **Table Name**.
2. In the **When output file exists** dropdown menu,
    - select **Overwrite** to substitute files with the same file names,
    - select **Skip** to stop the export operation from overwriting existing data.
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** checkbox.

Additional options for Microsoft Power BI file types

1. Enter your **Table Name**.
2. In the **When output file exists** dropdown menu,
    - select **Overwrite** to substitute files with the same file names,
    - select **Skip** to stop the export operation from overwriting existing data.
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 SAS Transport Format and SAS V8 Transport Format file types

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

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.
    - select **Skip** to stop the export operation from overwriting existing data.

Additional options for XML file types

1. In the **When output file exists** drop-down menu,
    - select **Overwrite** to substitute files with the same file names.
    - select **Skip** to stop the export operation from overwriting existing data.
2. Select the **Legacy compatibility mode – Ignore format customizations during export** checkbox if you want to use the Monarch Classic style formatting instead of the column format customizations during the export of numeric and date columns.

> **NOTE**: When this option is selected, numeric values are exported using the current system's localized decimal separator. Date values are exported in "yyy-mm-ddThh:mm:ss" format. For Monarch Server, this would typically be based on the Service Account for the Monarch Server Agent Service localization specifically (or, if running Monarch Server in console mode, the User Account context in which the console is running).

Plug-ins supported by Automator

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 processed by all elements after the Hub separately **NOTES**: 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 ("Configuring Servers") in the Monarch Server Installation Guide. • 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:** 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 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 disabled if the model is not specified or does not have any summary. • **Name** — name of the graphic element. • **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. • **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 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.

Advanced options

Advanced options for PRF

1. **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.
2. **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 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 the data link was successful (i.e., to test whether you can connect to the specified OLE DB, ODBC, or Snowflake 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)

- 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)
- SAS Transport File Format V8 2xpt (*.2xpt)
- SAS Transport File Format V8 xpt (*.v8xpt)
- SAS Transport File Format V8 xpt8 (*.xpt8)
- Tableau Data (*.csv)
- Tableau Data Extract (*.hyper)
- 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.

From Monarch Server 2022.0, the Panopticon Designer export format (.dwx) has been deprecated, and processes with this file type from previous versions of Monarch Server are no longer supported. Running such processes will lead to a failed export error. Choose an alternative file type in this case.

- **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 data:** 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 if 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 if export file is XLS/XLSX)

The following options are available for Summary Exports:

- **Security:**
  - **Digital Signature**: This option is available for Table and Summary exports and allows you to embed the currently selected digital signature to the exported Excel file (XLSX/XLSM only). By adding a digital signature, you establish the authenticity, integrity, and non repudiation of the export output. Select or change the

certificate to be used by clicking the **Certificate** button. In the **Select Certificate** dialog box, specify the certificate source by choosing between *Store* and *File* in the dropdown menu.

    - Store: A list of the certificates currently available on your Certificate Store is displayed. Select a certificate from the list and click OK.
    - File: Navigate to the folder containing your certificate file and then select it. Enter the certificate password in the field provided. Click the Get Certificate Information button to display the details of the certificate.

- **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.
- **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 option is 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/XLSM Table Exports:

- **Security:**
    - **Digital Signature**: This option is available for Table and Summary exports and allows you to embed the currently selected digital signature to the exported Excel file (XLSX/XLSM only). By adding a digital signature, you establish the authenticity, integrity, and non repudiation of the export output. Select or change the certificate to be used by clicking the **Certificate** button. In the **Select Certificate** dialog box, specify the certificate source by choosing between *Store* and *File* in the dropdown menu.

        - Store: A list of the certificates currently available on your Certificate Store is displayed. Select a certificate from the list and click OK.
        - File: Navigate to the folder containing your certificate file and then select it. Enter the certificate password in the field provided. Click the Get Certificate Information button to display the details of the certificate.

- **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 if export file is HYPER)

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**: Select this option to use a password file in .txt format.
- **Password**: The password to the Tableau Server (**NOTE**: If the **Use Password File** option is selected, you need 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 source with the data from the export table (**NOTE**: Date modified is changed);
    - **Replace data**: Replaces the data in the data source with the 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.

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). 4. Click the **OK** button. ◦ **Test** — use this button to check file availability in the specified location.
Check Cmd	Command line prerequisite that 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.	• **Name** — name of the graphic 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). 4. 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 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	• **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.
	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.	• **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 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** • **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 Definitions
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 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 are specified in the Global Scripts of Standard Processes, are imported in the 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, Amazon S3, Microsoft Azure, Oracle Cloud Infrastructure, and Google Cloud Storage login credentials for input, model, and export files.	• **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 **Use current user credentials** checkbox. • **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. From the dropdown menu, select the region endpoint. • **Microsoft Azure credentials** — specify the account name and account key in their respective fields. Click the **Set Active Directory Credentials** button to open Microsoft's sign in portal and input your credentials. To update or remove these credentials, click the **Update Active Directory Credentials** (i.e., this button is activated after a successful authorization) and **Clear Active Directory Credentials** buttons, respectively.

		NOTE: Objects can also be imported from and exported to *Azure Data Lake Storage Gen2* and *Azure Blob Storage* via Azure Active Directory authorization. • **Oracle Cloud Infrastructure credentials (S3 compatible)** — specify the access key, secret key, and endpoint URL in their respective fields. • **Google Cloud Storage credentials (S3 compatible)** — specify the access key and secret key in their respective fields. **NOTE**: Select the **Use Defaults defined in Server Settings** to use the credentials saved from the Server Integration tab in System Preferences.
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.	• **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

	• 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.	• **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 two processes are connected by a red connector.	• **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
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 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 Definitions
Log	Allows you to create an event written to 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.	• **Type** — Select the type from the 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. 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 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. In the **When destination files exist** option:

   - Select **Raise error** to receive an alert of an existing file.
   - Select the **Overwrite** check box 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. For multiple input files, selecting this checkbox and clearing the **One zip per file** checkbox archives all files in one compressed file. The **Zip Options** settings also display.

   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 store directory path.
   - Select the **Set password** check box to set the password.

6. Select the **One zip per file** checkbox if you want each input file to be archived into its own compressed 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 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:** Click the **Recipients** button to set the recipients of the e-mail based on keys. **NOTE**: Clicking the **Recipients** button opens the **Edit Recipients** dialog box. From this dialog box, you can view, add, modify, or delete e-mail recipients. You can also import keys and e-mail addresses from a CSV file through the *Recipients file path* field. Click the Browse icon to select the desired CSV file and then click either **Import with append** or **Import with replace**, whichever is appropriate, to proceed with the import. The list should now be populated with the keys and e-mail addresses from the imported CSV file. 3. **Missing key recipient:** Allows you to set the e-mail address to which missing keys will be sent. 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. 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. For multiple input files, selecting this checkbox and clearing the **One zip per file** checkbox archives all files in one compressed file. The **Zip Options** settings also display. **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 password to the archived file. 6. Select the **One zip per file** checkbox if you want each input file to be archived into its own compressed 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 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 expose 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: Boolead

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** – 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** – Click to manually perform mapping or modify existing mapping.
- **Delete Mapping** – Click to delete currently selected mapping.
- **Delete All Mappings** – 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 .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 Visual 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**, or **Cron**.
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 please 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
  - Building executing plan
  - Ready to run
  - Running
  - Completed
  - Some items completed with errors
  - Some prerequisites completed with errors
  - Some exports completed with errors
  - Canceled
  - Cancelling
  - Process awaiting retry after a failed run attempt
  - Dispatcher host is not responding (a process has been executed on a host that is currently not responding)
  - Orphan job (a process has been executed on a host that does not exist in the current configuration)
  - Cancelled automatically (a process log is stuck because the host became unavailable and another log was created after the host became available)
- The name of the server on which the process is run
- DataPrep version

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:

Symbol	Description
~	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 (XML)** button to export the log to an XML file.

Click the **Export Log (HTML)** button to export the log to an HTML file.

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

Click the **Go to Process** button to navigate to the Process details page (i.e., the process that was run and created in the current log).

Click the **Previous**/**Next** button to navigate to the previous/next log from the Logs list.

Alerts

In the latest version of Automator, you can now define an alert for Visual Processes. Three predefined alerts can be used, and any number of additional alerts can be defined using scripting.

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

Use this page to add, edit, or delete alerts.

Adding an Alert

Alerts allow users to be notified based upon certain conditions. You can, for example, be notified via email if a process fails to run. These notifications are called alerts. Some alerts are pre-defined within Automator itself, such as the process failure (SystemAlertProcessFailed) and process element failure (SystemAlertProcessElementFailed) alerts. Others can be defined by users in 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.

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 SystemAlertProcessFailed) 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 (except for the SystemAlertProcessElementFailed).

4. In the **Description** field enter alert description.

NOTE: If you use the SystemAlertProcessElementFailed alert, you can choose one or more element types by clicking .

5. If the alert should be immediate, check the **Rise Alert Immediate** checkbox.

NOTE: The **Rise Alert Immediate** checkbox is available only for the SystemAlertNoInputToExporter alert and the SystemAlertProcessElementFailed alert.

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 .
  - Type the message subject in the **Subject** box.

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

  - Type the message in the Message 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 level from the following: **None**, **Medium**, **Maximum**.
    - Select the **Store directory path** check box if necessary.
    - Select the Set **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 folder by clicking the **Browse** button. The **Select Folder** 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 Paths** box.
    - Click **Ok** to add the folder.
  - 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 alert. 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** section 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.

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 toInserting 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.
- **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.

Guidelines for Valid and Invalid Characters

This section presents important guidelines for the characters used in VBScript.

VBScript supports all ANSI characters (ASCII values 0–255) in strings. You can include these characters using the following:

- Their literal form (if the keyboard or encoding supports it).
- The Chr() function, which converts an ASCII value to its corresponding character.

A complete list of ASCII values from 0 to 255, including their decimal values and corresponding characters and descriptions, can be viewed here.

NOTE: ASCII values from 0 to 127 are standard ASCII, and ASCII values from 128 to 255 are extended ASCII (specific to certain code pages, e.g., Windows-1252).

If you're dealing with extended ASCII characters (128–255), these characters may be valid in VBScript strings, but their behavior depends on the system's character encoding. If the system uses a different code page or non-ANSI encoding, characters may not display as expected. For example, You can use characters like ©, ®, or é, but be mindful of encoding when reading/writing files or interacting with external systems.

For testing, string lists are as follows:

Category	Lists
Control Code	N/A
Space	
Basic Symbols	!"#$%&'()*+,-./
Digits	0123456789
Digit Separators	:;<=>?@
Uppercase Letters	ABCDEFGHIJKLMNOPQRSTUVWXYZ
Uppercase Adjacent	[\]^_`
Lowercase Letters	abcdefghijklmnopqrstuvwxyz
Miscellaneous	{|}~
Extended	ÇüéâäàåçêëèïîìÄÅÉæÆôöòûùÿÖÜø£Ø×ƒáíóúñÑªº¿®¬½¼¡«»░▒▓│┤ÁÂÀ©╣║╗╝¢¥┐└┴┬├─┼ãÃ╚╔╩╦╠═╬¤ðÐÊËÈıÍÎÏ┘┌█▄¦Ì▀ÓßÔÒõÕµþÞÚÛÙýÝ¯´±‗¾¶§÷¸°¨.132■

The script dialog should allow for all of the above to be used in comments but not in string names per VBScript syntax rules (which will result in an appropriate VB error accordingly).

Valid string (variable) names in VBScript must use the following:

- Letters (A-Z, a-z)
- Digits (0-9), but not as the first character
- Underscore (_), but not as the first character

The guidelines for variable names are as follows:

- Control characters (ASCII 0–31) are generally invalid in strings.
- Use two double quotes ("") to include a quote inside a string.
- Single quotes within strings are acceptable as long as they are part of the string.
- Backslashes do not need escaping in VBScript.
- Variable names cannot include special characters such as spaces or punctuation.

When handling control characters, they can be referenced using their ASCII values (e.g., Chr(10)), but care must be taken with their behavior in strings.

VBScript does not have a built-in syntax for multi-line comments. However, you can simulate them by starting each line with a single quote or Rem.

Connection Definitions

The **Connections 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: OLE DB for Native MS SQL Provider, OLE DB Provider, ODBC Provider, or Snowflake Native Provider for Export.
     b. Provide parameters according to provider type chosen.
     c. Click the **Test Connection** button to test whether the data link was successful (i.e., to test whether you can connect to the specified OLE DB, ODBC, or Snowflake 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.

   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.
5. 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
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. This option becomes available when a summary is selected for export. For multiway export by tables, if all summaries are selected, the system will apply the summary name for the table name by default. As for multiway export by files, no extra changes are needed because the file name is set to the summary name. Note: This macro is not available when all or more than one load plan is selected.
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. Note: This macro is not available when all or more than one load plan is selected.
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 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 A?a??a?? counts across all jobs in all processes.

Load Plan		&[loadplan]	Name of load plan selected.
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.
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
Time Intervals	Calendar Year	&[yyyy]	4-digit calendar year
	Calendar Half-year	H&[halfyear]-&[yy]	The calendar half-year in which the date falls, where each half-year is defined as six calendar months.

	Calendar Quarter	Q&[qtr]-&[yy]	The calendar quarter in which the date falls, where each quarter is defined as three calendar months
	Calendar Month	&[monthname]-&[yy]	The calendar month
	Calendar Week	W&[week]-&[yy]	The week number
	ISO8601 Year	&[isoyear]	The year corresponding to the ISO week in which the date falls.
	ISO8601 Week	&[isoyear]-W&[isoweek]	The ISO week in which the date falls.
	Fiscal Year	FY&[fendyear]	The fiscal year expressed as the calendar year of the first day of the fiscal year in which the date falls
	Fiscal Half-year	FH&[fhalfyear]-&[fendyy]	The fiscal half-year in which the date falls
	Fiscal Quarter	FQ&[fqtr]-&[fendyy]	The fiscal quarter in which the date falls.
	Fiscal Period	FP&[fperiod]-&[fendyy]	The fiscal period; either a calendar month or a 4- or 5-week period, depending on the type of fiscal year
	Fiscal Week	FW&[fweek]-&[fendyy]	The fiscal week number in which the date falls.

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
&[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.
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.

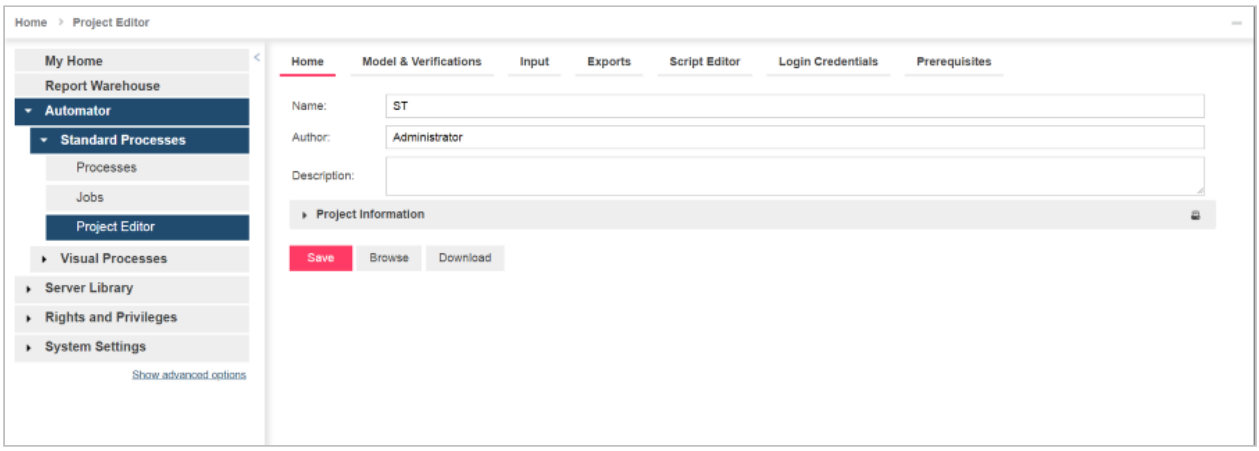
Other than XML, the following JSON structure may also be used:

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

```
}

]

}
```

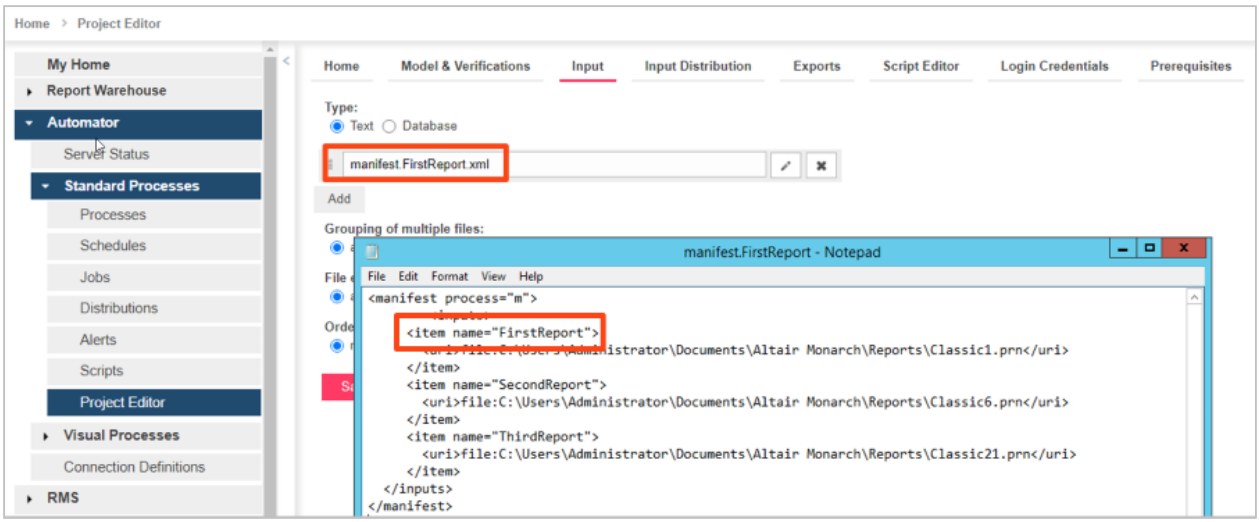
To proceed with manifest monitoring

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

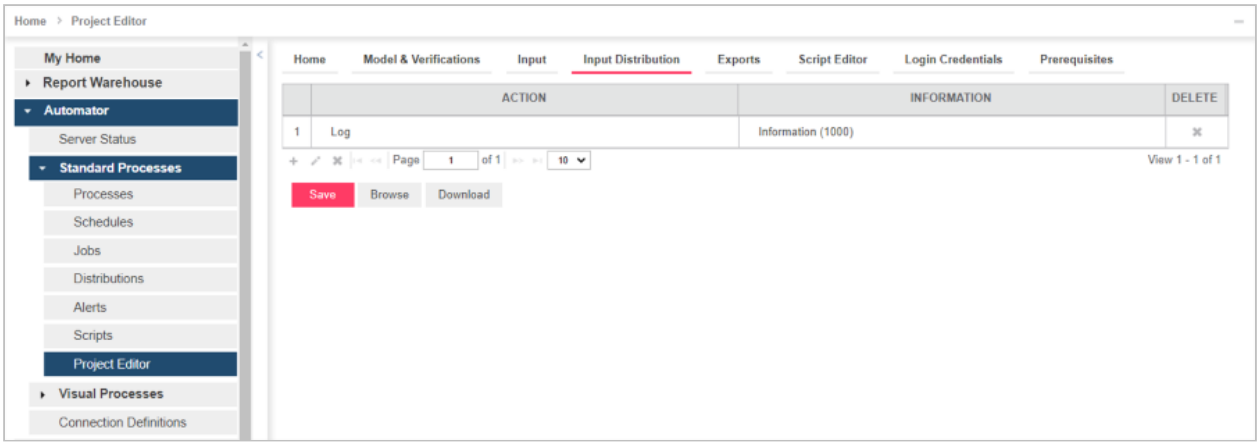


2. Open the **Model & Verifications** tab, and set the model for the new project. See [Specifying Model Settings for a Project File](#).

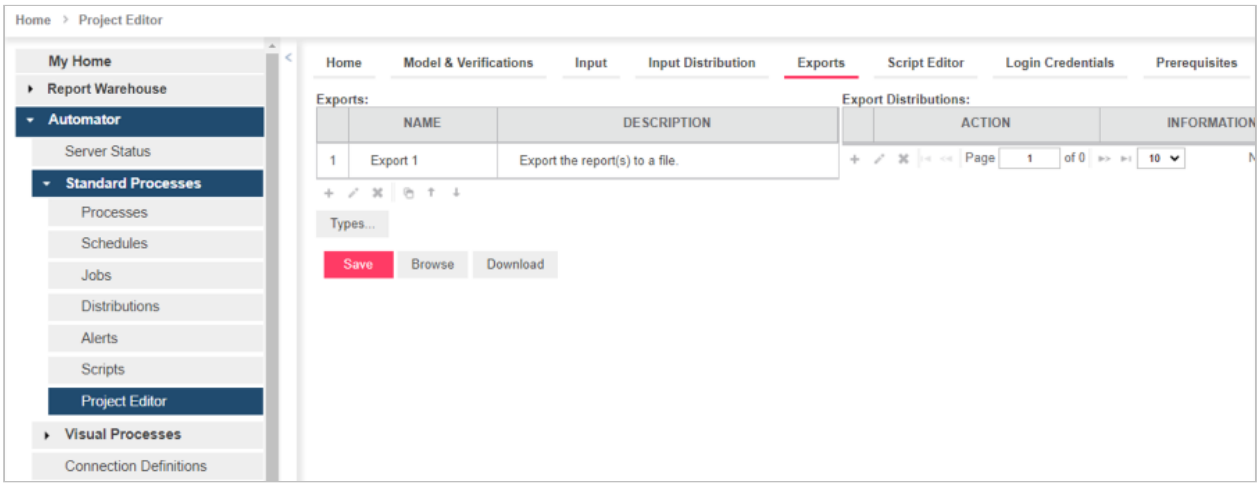
3. Open the **Input** tab to add an input file. If you would like to use an xml file as the input file, rename the file as "manifest.xxx," where "xxx" is the item name from the xml file.



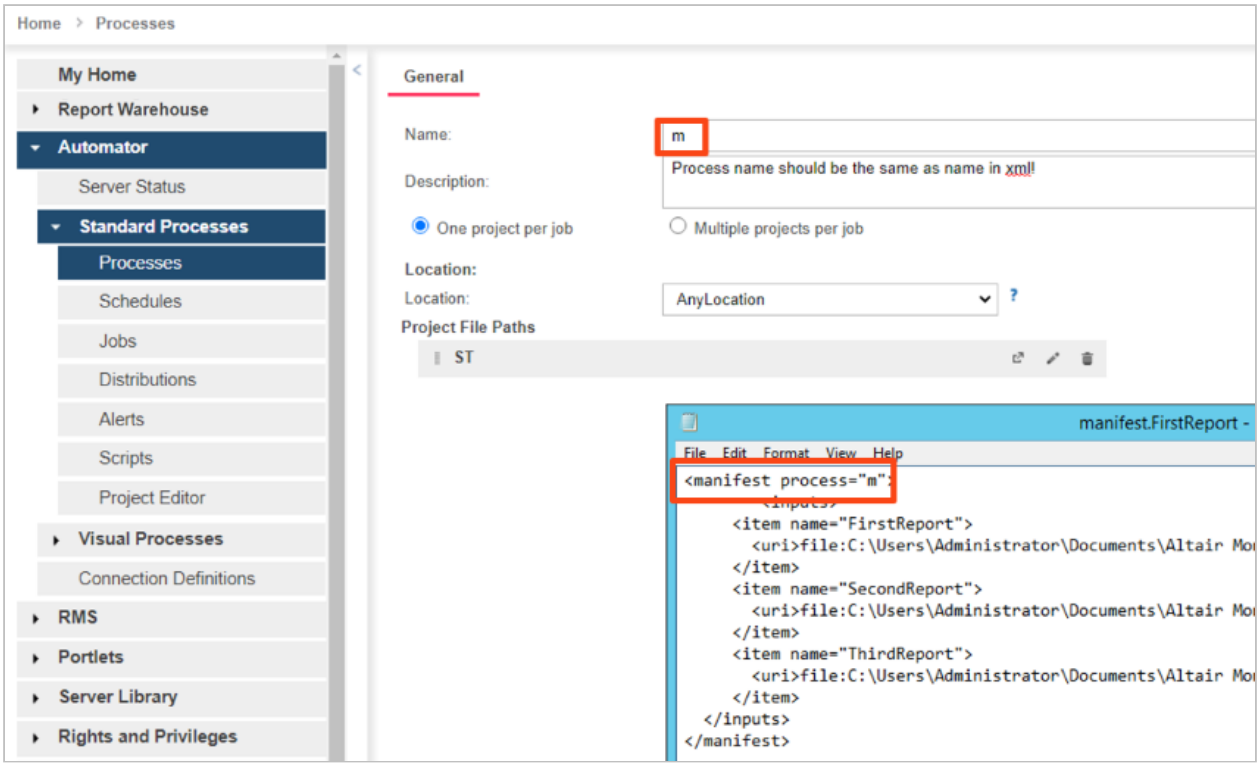
4. Add any Input Distribution.



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**.

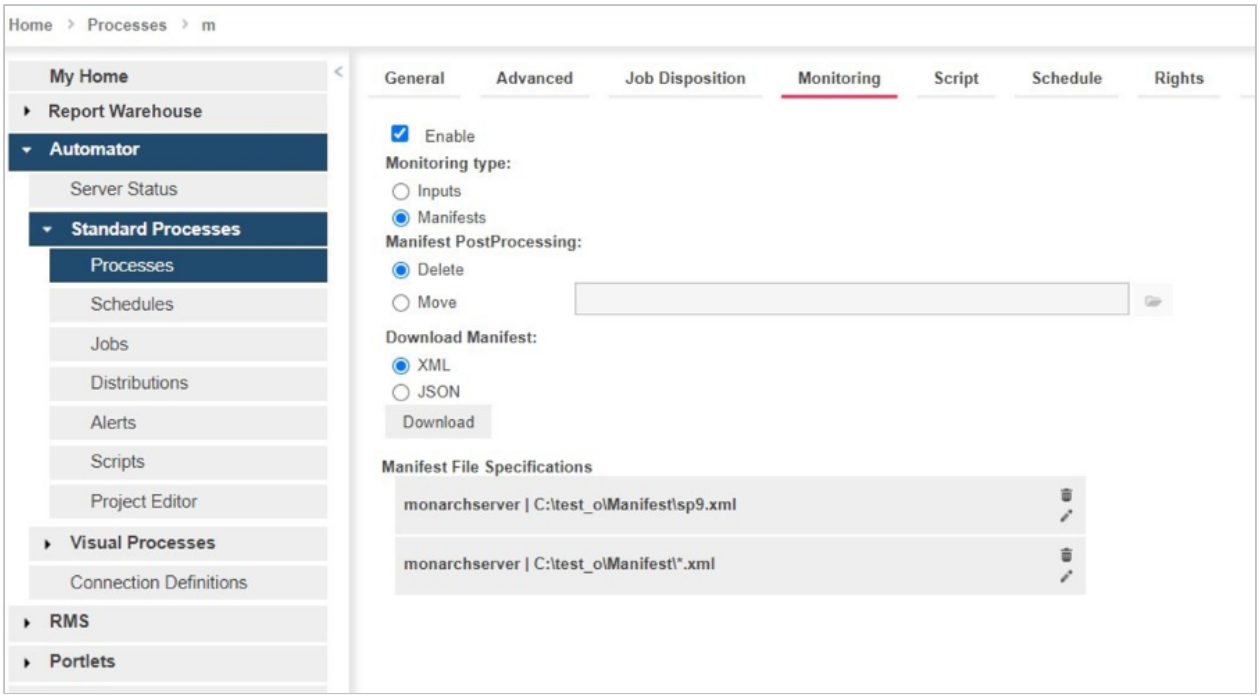
> **NOTE**: The process name in the XML file (“m” in the example above) should be the same as the name of the Standard Process created.

8. Open the process you created in Step 6.

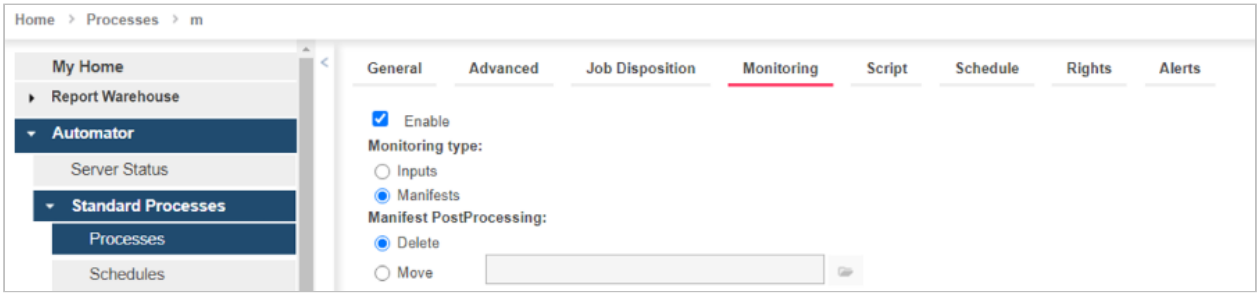
   In the **Monitoring** tab, click the **Enable** check box, and select **Manifests** under **Monitoring Type**.

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

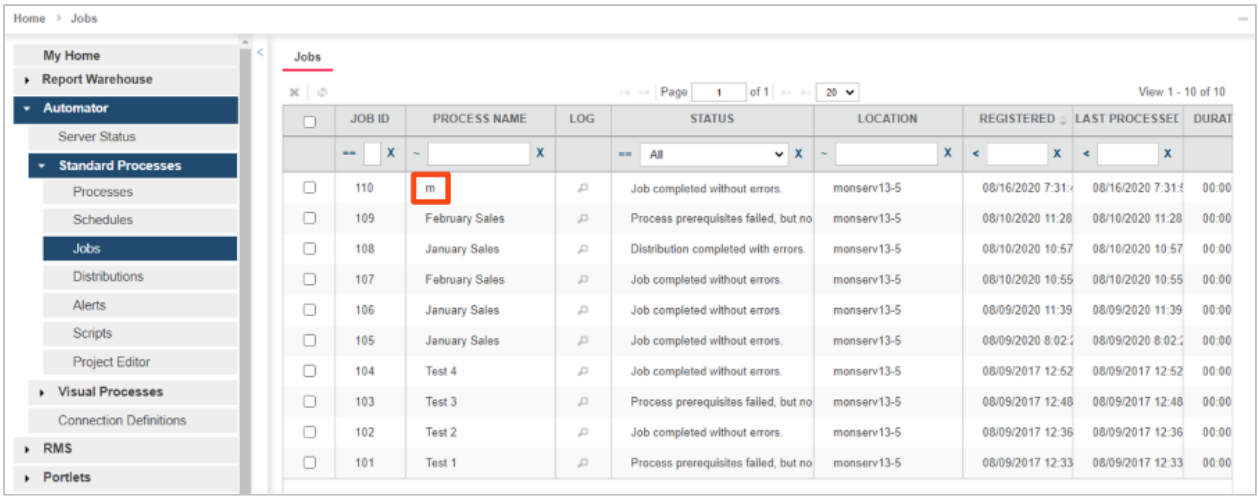


11. **Save** the process.
12. Proceed to the **Jobs** page under **Standard Processes** to view the job log or re-run the job.

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.

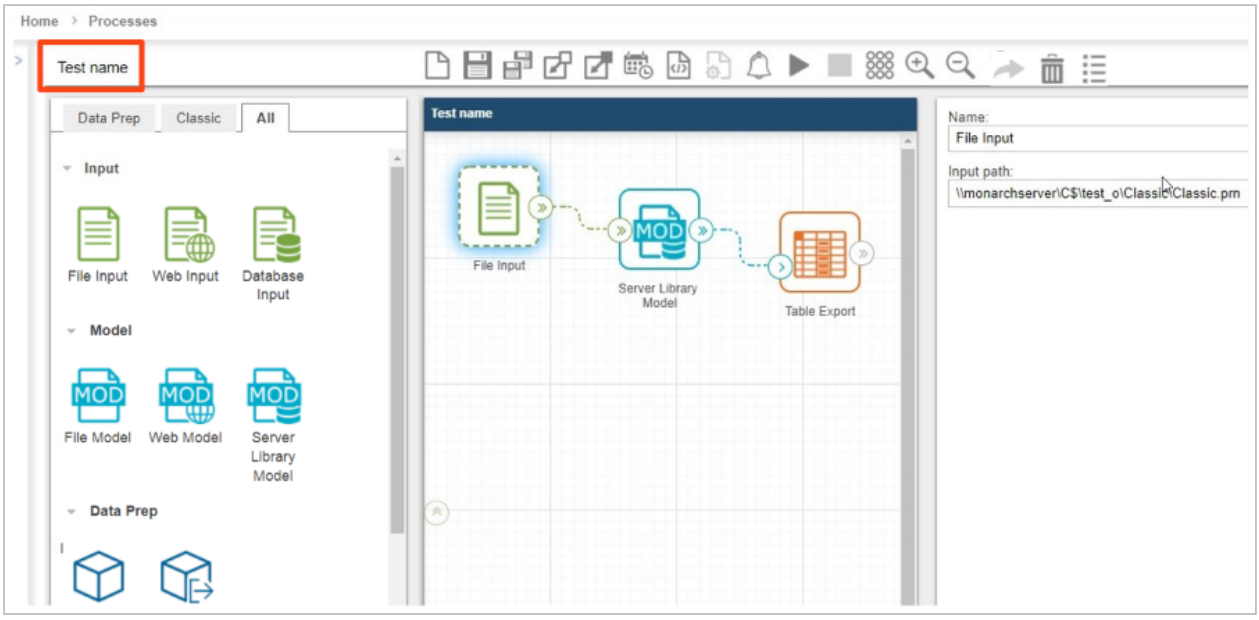
The following JSON structure may also be used:

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


"ProcessItems": [
    {
      "ItemType": "InputFile",
      "FileInput": {
        "FileLocation": "Input path",
        "Name": "File Input",
```

```
"PdfPassword": "password"

        }

      }

]

}
```

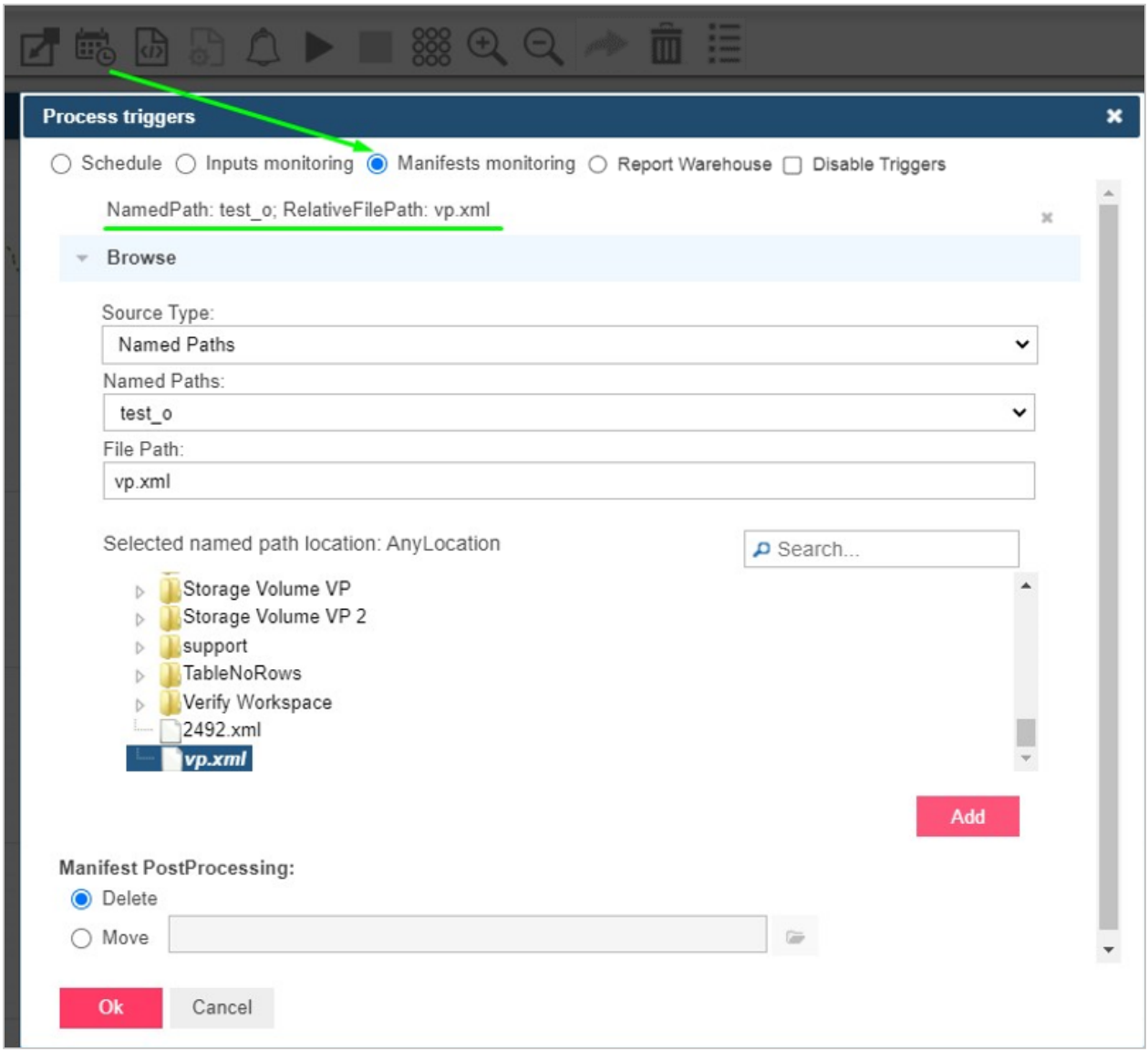
To proceed with manifest monitoring

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**.

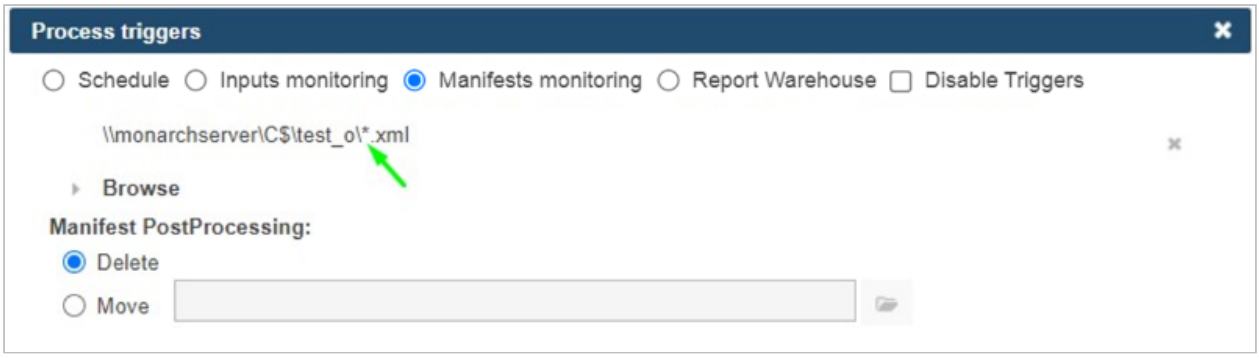


NOTE: You may use a wildcard (* or ?) to select any file (e.g., *.xml).

d. Select one of two options in Manifest PostProcessing.

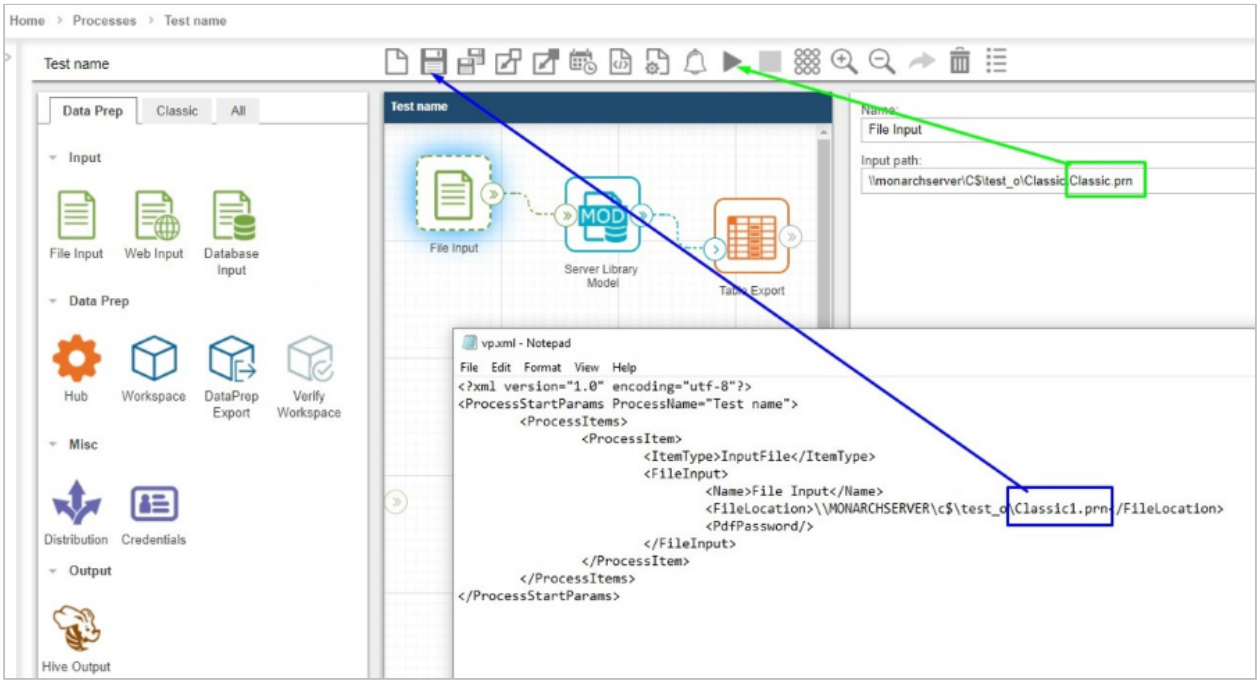
   - Delete (selected by default) for remove the XML file after processing
   - Move, set the folder path, for move the XML file after processing

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



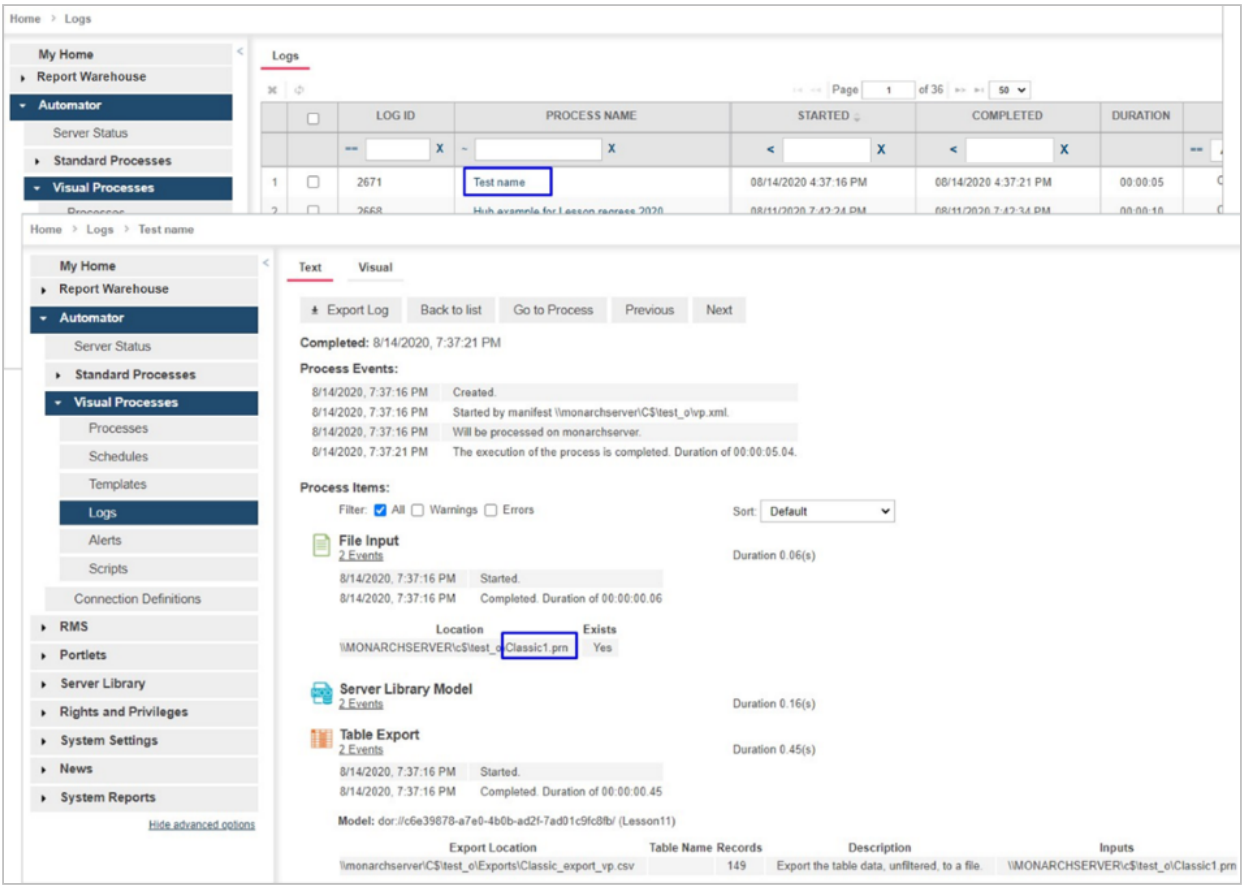
3. Save the process.

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.



4. Close the process, and open **Logs**.

   The log list should contain the process that will run.



5. 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 name="TableName" type="xs:string">
                <xs:element name="WhenTableExist" type="ExportTableExistence"
nillable="true">
                <xs:element name="Destination" type="xs:string">
              </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: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: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>
```

Server Library

The topics in this section provide information about:

- Folders
- Project Files
- Workspaces
- Change Lists
- Models
- Bulk Edits

Folders

You can create server library folders that include particular project and 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

NOTE: The options for **Projects**, **Workspaces**, and **Change Lists** are only available for Automator. The option for **Templates** is only available for Report Warehouse and Report Mining Server.

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.

NOTES:

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 the 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.

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.

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 Server** 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. Edit the necessary parameters in the dialog box that displays.

      - For workspaces 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 workspaces based on OLEDB, e.g., Connection Name, SQL Dialect, Timeout.
      - For workspaces based on ODBC, e.g., ODBC DSN, User ID, and Password.
      - For workspaces based on Google Analytics, e.g.,Key File Path (.json format), Property ID, Categories, Dimensions, Metrics, and Selected Columns.
      - For workspaces 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:

   Click the name of the workspace you want to edit or click anywhere on the row, and then click .
3. Make the necessary changes. For details about how to change workspace parameters refer to Adding a Workspace topic.
4. 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 change list, 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, 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 the entire 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 can be modified via Bulk Edits:
    - **Alerts**: Description, Email, Log, Rss
    - **Distribution** (Visual Processes): Distribution path, Copy, Move, Translate, Email, Log, and RSS.
    - **Models**: Name, Description, External model path, External lookup data source
    - **Projects**: Name, Description, Model, Inputs, Exports, Scripts, Input distributions, Export distributions

- **Templates**: Content
- **Workspaces**: Name, Description, Input filename, OLE DB settings, SQL Server settings
- **Change Lists**: Name, Description
- **Visual Processes**: Monitoring, Database input, File input, Web input, DataPrep export, Table export, Summary export, Report export, Workspace, Distribution, Scripts
- **Standard Processes**: Monitoring, Export Behavior for accdb/dbf/mdb, Scripts

- For scripts for Standard and Visual Processes, the following options for Bulk Edits 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
- Script source

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 manage roles
- How to manage user groups
- How to manage users

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.
- **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.

        - **Can Use Default Credentials**: Allows the user to use the credentials defined in the Server Integration settings.
    - **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.

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 at the top of the page to specify appropriate rights. The following tabs are available:
    - **Models:** Assign models to the current user.
    - **Automator Processes:** Grant access to Automator processes for the role, user group, or user.
    - **Visual Processes:** Grant access to Visual Processes for the role, user group, or user.
    - **Automator Projects:** Grant access to Projects for the role, user group, or user.
    - **Workspaces:** Grant access to Workspaces for the role, user group, or user.
    - **Change Lists:** Grant access to Change Lists for the role, user group, or user.
    - **Named Paths**: Assign named paths to the current role, user group, or user.
    - **Folders:** Assign folders to the current role, user group, or user.
    - **Connection Definition:** Assign available connection definitions to the current role, user group, or user.
4. Assign the appropriate rights.
5. To save changes, click **Save**.
6. Repeat steps 3-5 until you have assigned all desired rights to a role, user group or user.

Setting Security Provider

A security provider provides the system with security information about user groups and users, as well as checks user passwords for validity.

To select the security provider of the system, on the Navigation Tree, click **Rights and Privileges**, and select **Security Provider**.

You can select one of the following options:

- **Active Directory**. The system is supported by Active Directory Authentication, which is based on (and integrated with) the security system of Windows NT.
- **Default**. The default provider presents an ordinary security model dealing with roles, groups and users, which are kept in the MS database. The default provider does not interact with security systems.
- **Strong password provider**. The Strong Password Provider supports the same security model as the default provider, but applies stricter rules to passwords. In particular it supports password history, length limitation and expiration period. If you select this option, you can specify the following settings:

- **Number of passwords stored in the history**: The valid values are integers in the range 0 - 100. This value specifies the number of last used passwords that the system remembers for each user. The system does not allow assigning a new password, if it is in the password history. Zero value disables this functionality. The default value is 15.
- **Minimum password length**: The valid values are integers in the range 0 - 128. The value specifies the minimum password length. Zero value disables this functionality. The default value is equal to 7.
- **Password age in days**: The valid values are integers in the range 0 - 365. The value specifies the number of days after which the password will expire. Zero value disables this functionality. The default value is 30 days.
- **Password must meet complexity requirements**: This check box enables validation of complexity requirements for the password. These requirements include the following rules: within the password string user must use at least three characters out of the four types of characters: digits (0-9), upper case alpha characters (A-Z), lower case alpha characters (a-z), and special characters which are pretty much the rest of the keyboard.
- **Force a new user to change the password at first logon**: If you select this check box, a user must change the password at the first logon, in case an administrator does not set a password for the user.

NOTE: All the settings of the Strong Password Provider are global and have effect for all users.

- **Show SSO**: Select this check box and the **Active Directory** option, to enable the user to sign in to the system by clicking the **SSO** link on the Login page, without having to type the name and password each time.
- **Enable Active Directory Group Lookup**: to allow searching and importing user groups on the Importing a User Group page.

Roles

Roles are used to set rights and privileges for users and user group. A role can include both user groups and users, but cannot include other roles. A role is kept in the database and has no tie with other security systems. An administrator can define rights via roles (Role Based Security) or assign them to user groups and users individually.

All roles used in the system are listed on the Roles page. To display the Roles page, on the Navigation Tree, click **Rights and Privileges** and select **Roles**.

Use this page to set rights and privileges for roles, as well as to add, edit, or remove roles from the list.

Adding a Role

Roles allow you to quickly assign specific access rights and privileges to a user or user group. Because access rights and privileges in Monarch Server are cumulative, when you add a user or user group to a role, the user or user group members are granted the role's access rights and privileges in addition to their own.

To add a role

1. On the Navigation Tree, click **Rights and Privileges**, and select **Roles**.
2. Click + on the **Roles** page. The **Role** page containing the following items appears:
    - **Role Name**: Enter a unique name of a new role.
    - **Description**: Optionally, enter any text defining the role.
    - **Privileges**: By selecting check boxes, an administrator can grant the following privileges:
        - **Administrator**: Allows the user to log on to MSAdmin.
        - **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.
            - **Can Use Default Credentials**: Allows the user to use the credentials defined in the Server Integration settings.
        - **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.
    - **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.

    - **Models** tab allows you to assign models to the current user. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
    - **Automator Processes** tab allows you to assign automator processes to the current role. To do this, select a search from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.
    - **Visual Processes** tab allows you to assign visual processes to the current role. To do this, select a search from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.

NOTE: Processes created in older versions of Monarch Server (e.g., v13.2) are assigned to all users.

    - **Automator Projects** tab allows you to assign projects to the current role. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
    - **Workspaces** tab allows you to assign workspaces to the current role. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.

- **Change Lists** tab allows you to assign change lists to the current role. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
- **Folders** tab allows you to assign folders to the current role. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
- **Named Paths** tab allows you to assign named paths to the current role. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.

NOTE: You can copy rights of a role by clicking the **Copy Profile** button available on the aforementioned tabs, except the **Views** and **Report Security** tabs.

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.
     - **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.
       - **Can Use Default Credentials**: Allows the user to use the credentials defined in the Server Integration settings.
     - **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.

    - **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 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.
   - **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.

   - **Models** tab allows you to assign models to the current user group. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
   - **Automator Processes** tab allows you to assign Automator processes to the current user group. To do this, select a search from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.
   - **Visual Processes** tab allows you to assign visual processes to the current user group. To do this, select a search from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.

     NOTE: Processes created in older versions of Monarch Server (e.g., v13.2) are assigned to all user groups.
   - **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**.

NOTE: You can copy rights of a user group by clicking the **Copy Profile** button available on the aforementioned tabs, except the **Views** and **Report Security** tabs.

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.
- **Force password change on next login**: Select this checkbox to prompt new users to change their password upon logging into the system for the first time. This option is selected by default only for new users.
- **Privileges**: By selecting check boxes, an administrator can grant the following privileges:
  - **Administrator**: Allows the user to log on to MSAdmin.
  - **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.
    - **Can Use Default Credentials**: Allows the user to use the credentials defined in the Server Integration settings.
  - **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.
    - **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.

    - **Models** tab allows you to assign models to the current user. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
    - **Automator Processes** tab allows you to assign Automator processes to the current user. To do this, select a search from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.
    - **Visual Processes** tab allows you to assign visual processes to the current user. To do this, select a search from the list, then drag appropriate items from the left pane to the right pane, and click **Save**.

      NOTE: Processes created in older versions of Monarch Server (e.g., v13.2) are assigned to all users.
    - **Automator Projects** tab allows you to assign projects to the current user. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
    - **Workspaces** tab allows you to assign workspaces to the current user. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
    - **Change Lists** tab allows you to assign change lists to the current user. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
    - **Folders** tab allows you to assign folders to the current user. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.
    - **Named Paths** tab allows you to assign named paths to the current user. To do this, drag appropriate items from the left pane to the right pane, and click **Save**.

NOTE: You can copy rights of a user by clicking the **Copy Profile** button available on the aforementioned tabs, except the **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.

System Settings

In this section you will learn how to change:

- System preferences
- Application settings
- Information to be displayed in the "About" dialog box
- Global user 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.
- **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.
- **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.
- **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.
- **Expand Administrator Tree**: Select this check box, if you want the MSAdmin Navigation Tree to be expanded to second-level navigation options by default.
- **Hide advanced content:** Select this check box to hide some advanced options in the Navigation Tree and object settings.

> **NOTE**: No matter whether the **Hide advanced content** check box is selected or deselected, you can hide non-mandatory items by clicking the **Hide advanced options** heading inside the given section. To display them, click the **Show advanced options** heading.

- **Implementation Name**: This optional name will be displayed on the Login page and on the **Welcome** line of the user interface.

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.

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

Settings	Description
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.
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 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.

Test Email	Click the button to test email settings.
Send notification to system administrators when module terminated unexpectedly	Select this check box if you want that admins to receive notification when the module terminates unexpectedly.
Send notification to system administrators before license expiration	Selecting this check box will notify system administrators with defined email addresses that the instance license expires in N days (as determined by the **Show alert before license expiration** setting in the **System Preferences** tab). The notification will be sent once per day. **NOTE**: A notification email will only be sent if the SMTP and email settings are correct (i.e., the email test is OK). This check box is disabled by default.
Send notification to system administrators when monitoring was disabled by an error	Select this check box to notify users with common Admin privileges whenever monitoring is disabled due to an error. The notification will be sent via the users' email addresses.

Security Settings

Settings	Additional Settings	Description
Permission Control	**Do not include Global option on Rights tab**	A new "All Users Group" will be created automatically. This new user group will include all existing users **NOTE:** Rights may be lost if the "All Users Group" is deleted. **Option**: 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 user name or password, on MSAdmin.

- **Save:** Saves application settings.
- **Clear:** Clears all the boxes.
- **Cancel:** Cancels the operation.

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.

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
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:

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.

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 whether 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 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.
        1. Select the task execution frequency from the following options: **One Time**, **Hourly**, **Daily**, **Weekly**, **Monthly**, or **Cron**.
        2. In the **At** box, set the time you want the process to run.
        3. In the **On** box, set the start date of the process.
        4. Select the appropriate timezone from the drop-down list.
        5. 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, the server settings need to be modified. Use the Server Integration page to configure the settings for the following:

SERVER	REQUIRED FIELDS
HTTP Credentials	• Username • Password
FTP Credentials	• Username • Password
Microsoft Azure	• Account Name • Account Key

	NOTE: Objects can also be imported from and exported to *Azure Data Lake Storage Gen2* and *Azure Blob Storage* via Azure Active Directory authorization. **Additional Steps:** 1. Click the **Set Active Directory Credentials** button. A new tab displaying Microsoft's sign in portal is opened. 2. Sign in using your credentials. Back in the Server Integration page, either of the following messages are displayed: • Successful authorization: *Login data have been successfully updated*. • Failed authorization: *Timeout for updating Login Data. Please try again*. 3. Click the **Update Active Directory Credentials** button to modify the cached credentials or the **Clear Active Directory Credentials** button to remove the cached credentials.
Amazon S3	• Access Key • Secret Key
Oracle Cloud Infrastructure	• Access Key • Secret Key • Endpoint URL
Google Cloud Storage	• Access Key • Secret Key
Microsoft Power BI API	• Client ID • Client Secret Key • Host Name • Port • Redirect URL For further details, click here.
Microsoft Azure Storage API	• Client ID • Client Secret Key • Host Name

	• Port • Redirect URL

To modify the server settings

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.
3. Modify the required server settings.
4. Click **Save** to apply the settings or **Clear** to start over.

Microsoft Power BI API Settings

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.

> **NOTE**: To get the Client ID for **Power BI**, click here.

- **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 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 URL**: It is the URL 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.

Power BI API Keys and Application Registration

> **NOTE**: The following steps were documented on May 16, 2022, using Power BI Service 13.0.18218.51.

The integration with Power BI is divided into two phases:

- **Phase 1: Application Registration**: In this phase, the Automator application is registered using the following tools:
  - Power BI Embedding Tool
  - Azure Active Directory

  The registration of the application will allow also allow you to generate a **Client ID**, which is required in the integration with Power BI.

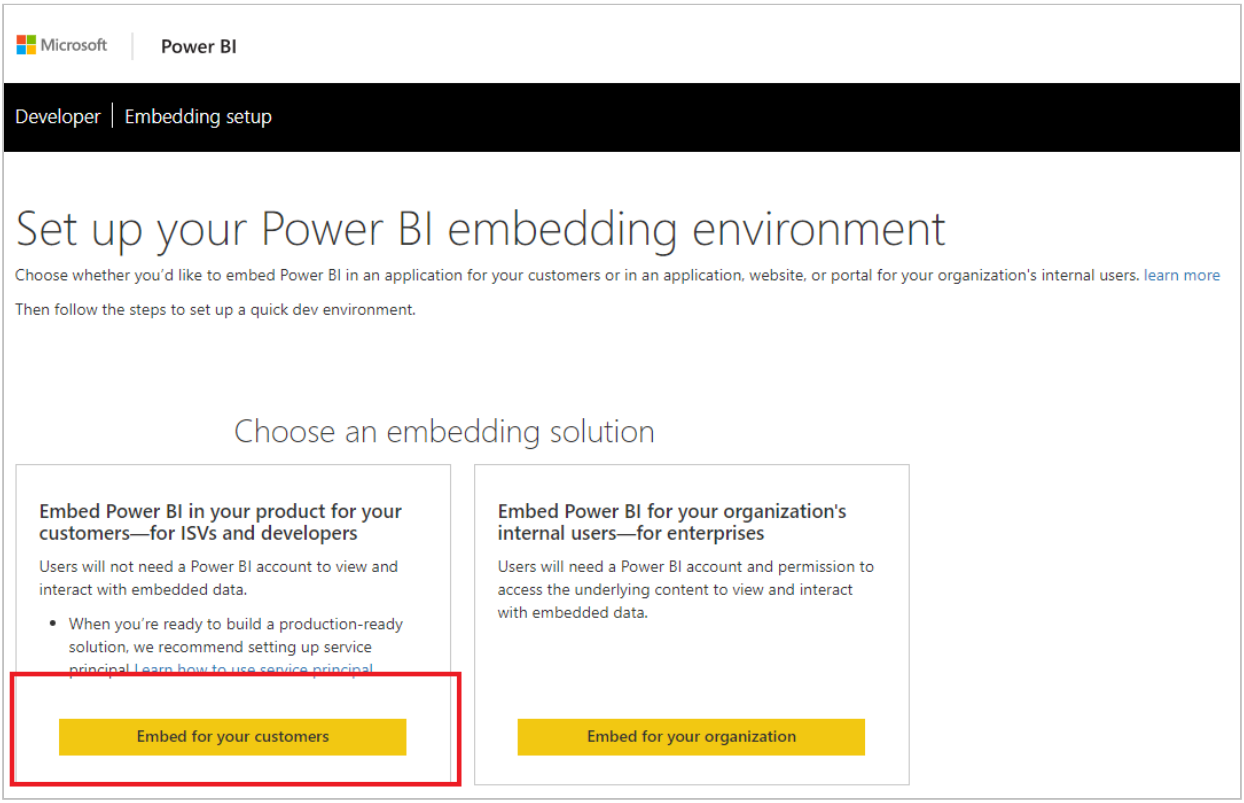
- **Phase 2: Generation of Client Secret Key**: After a successful registration in Phase 1, you can then generate the Client Secret Key via Microsoft Azure.

The related steps for Phase 1 and Phase 2 are presented below.

PHASE 1

To register the Automator application using the Power BI Embedding Tool

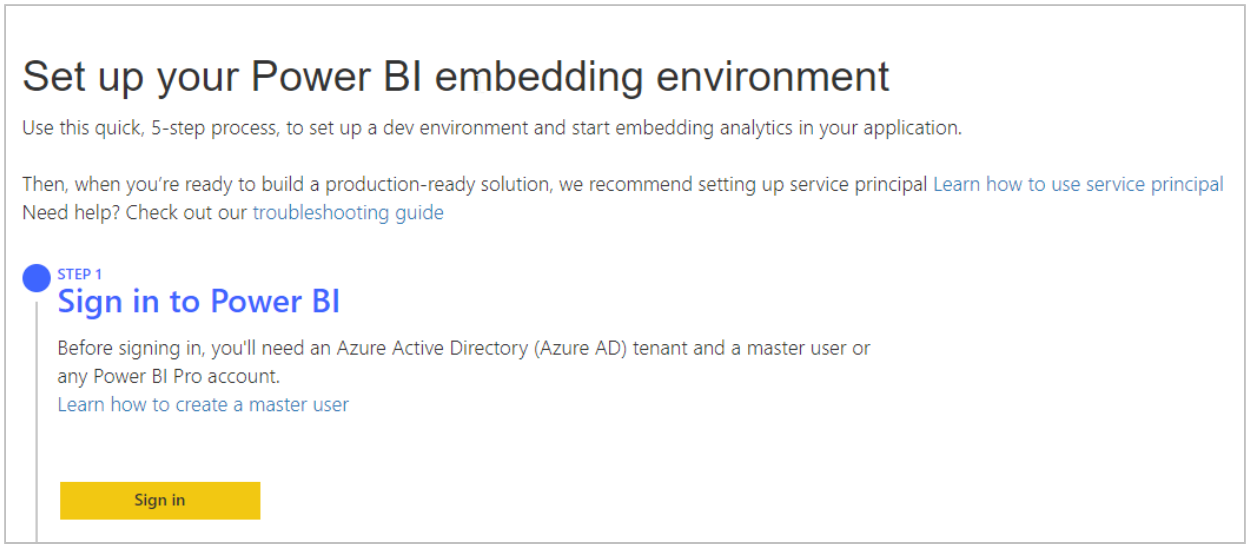
1. Access https://app.powerbi.com/embedsetup. From the choices of embedding solution, select **Embed for your customers**.



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.

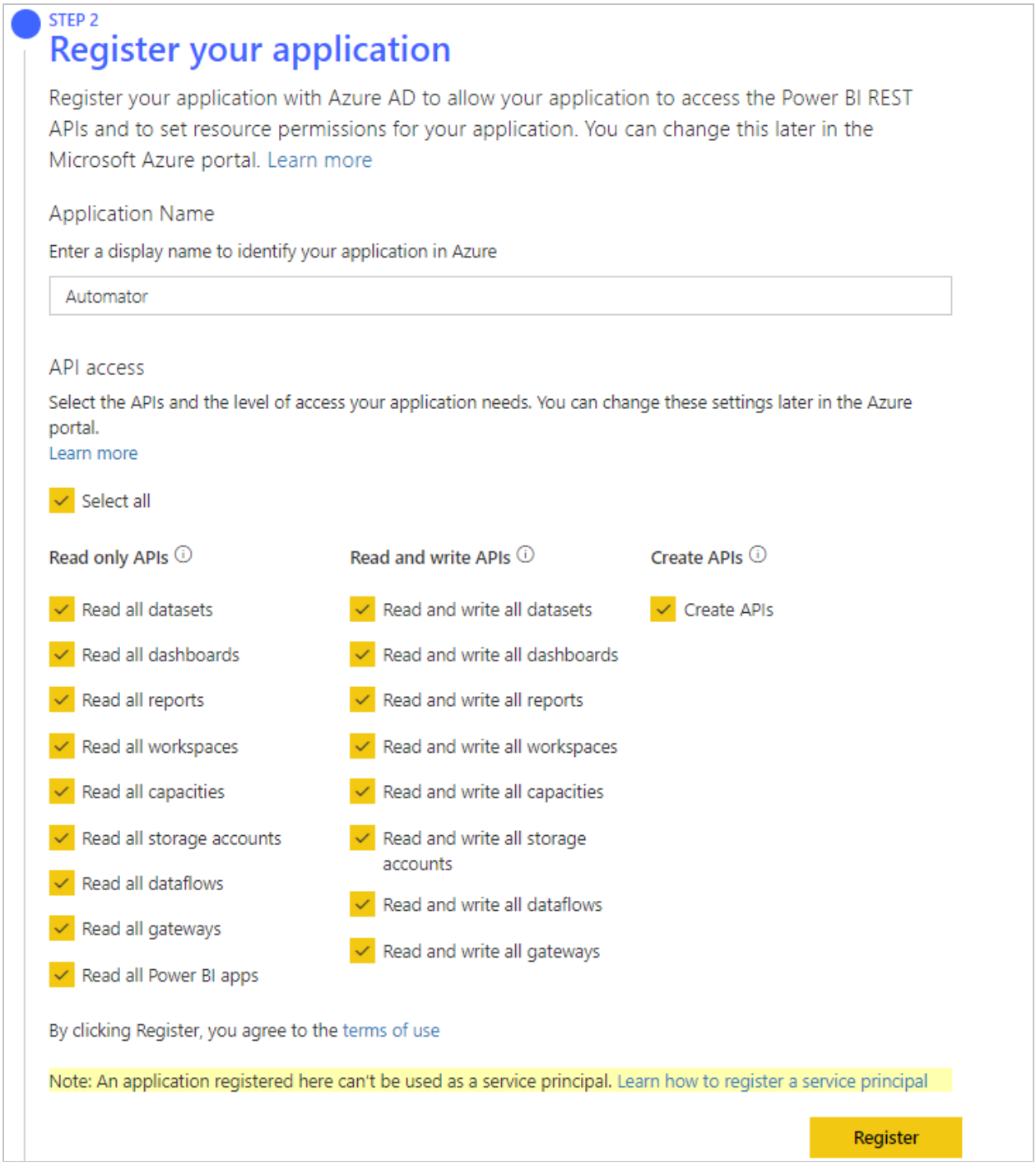
2. Sign in using your Power BI account in Step 1 of the **Embedding Setup** tool, and then click **Next**.



3. In **Step 2: Register your application**, provide the following details:

   a. **Application Name** – The name of your app, i.e., Automator.

   b. **API access** – Select the APIs you need as well as the level of access required. For details about permissions in Power BI, click here.



Click **Register**. You may opt to skip Steps 3 and 4 of the Embedding Setup tool at this point.

4. In Step 5 of the **Embedding Setup** tool, click **Grant permissions**. This shows the permissions requested for the application.

5. Click **Accept**.

Permissions requested

Automator2021
unverified

This application is not published by Microsoft.

This app would like to:

- View all datasets
- View all dashboards
- View all reports
- View all workspaces
- View all capacities
- View all storage accounts
- View all dataflows
- View all gateways
- View all Power BI apps
- Read and write all datasets
- Read and write all dashboards
- Read and write all reports
- Read and write all workspaces
- Read and write all capacities
- Read and write all storage accounts
- Read and write all dataflows
- Read and write all gateways
- Create content
- Sign you in and read your profile

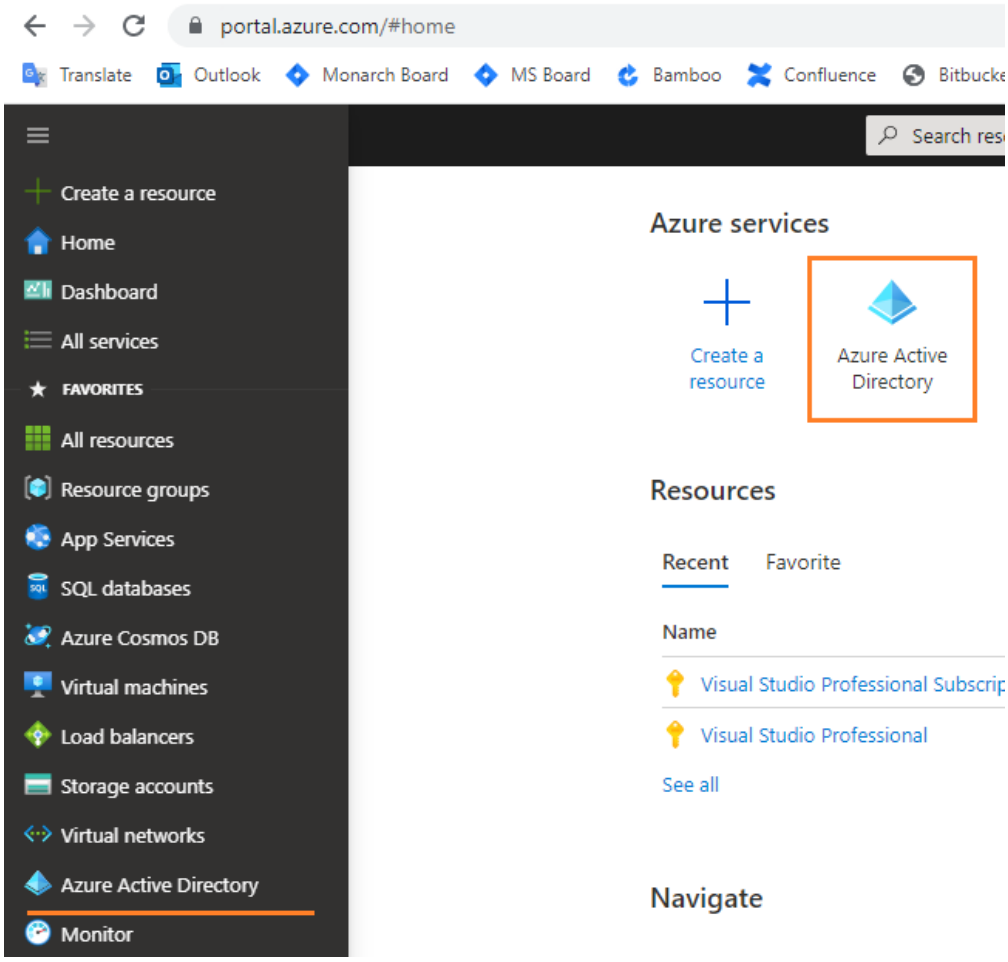
Accepting these permissions means that you allow this app to use your data as specified in their terms of service and privacy statement. **The publisher has not provided links to their terms for you to review.** You can change these permissions at https://myapps.microsoft.com. Show details

Does this app look suspicious? Report it here

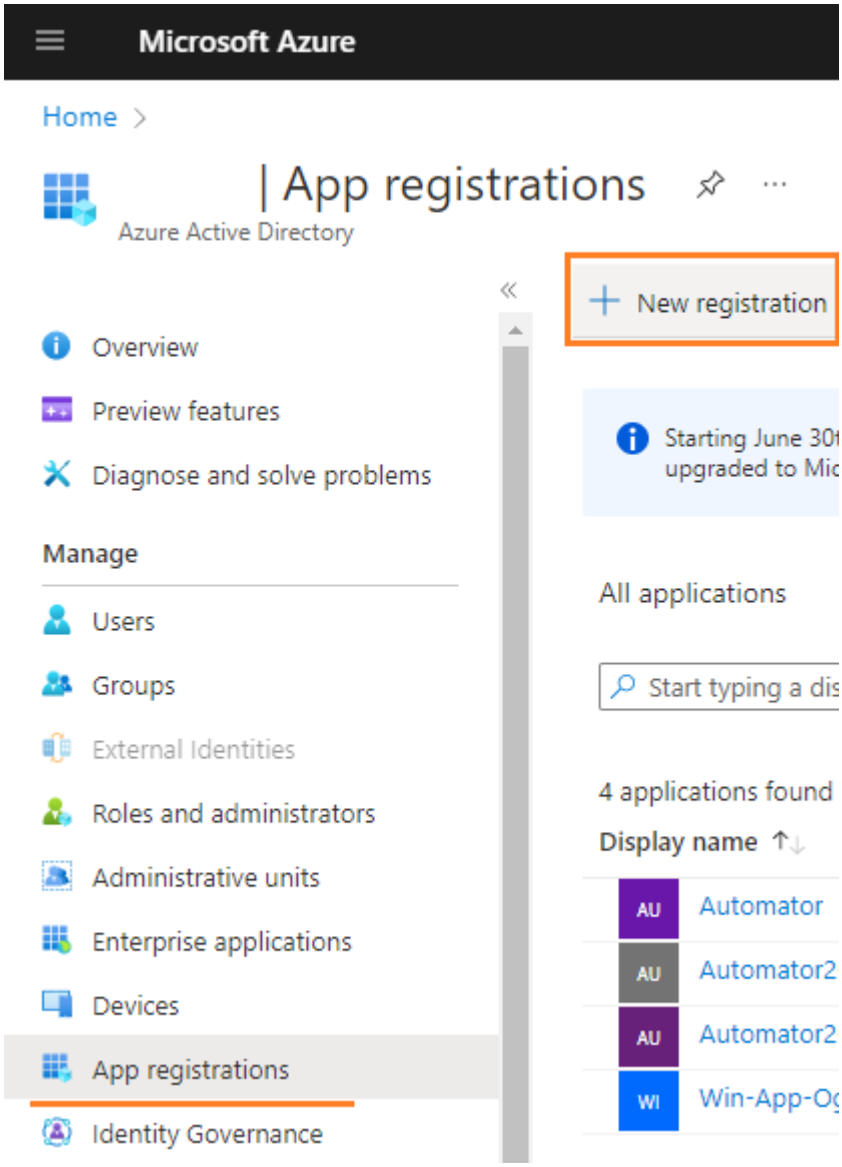
Cancel Accept

To register the Automator application using Azure Active Directory

1. Open the Azure Portal at https://portal.azure.com.
2. In the list of Azure Services, click **Azure Active Directory**.



3. Navigate to **App registrations** and click the **New Registration** button. Proceed with the registration of your application.



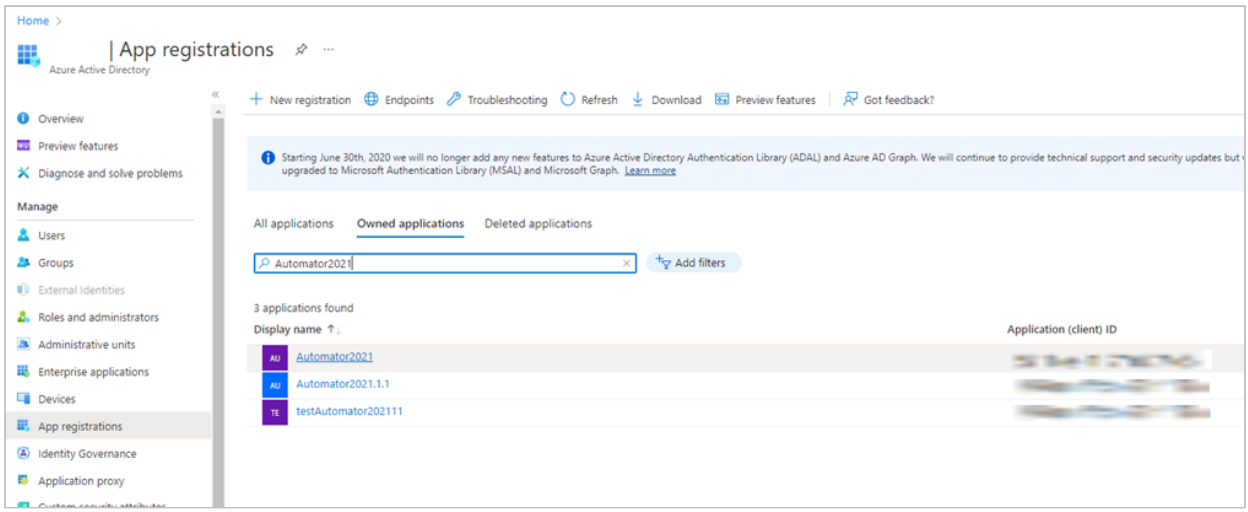
4. Navigate to page of the registered application by clicking its corresponding name from the list of App registrations.
5. Configure the application's API permissions. In the application's page, click **API permissions** from the left panel, and then click the **Add a permission** button.
6. Navigate to the Power BI Service, and click Delegated permissions to request API permissions.

PHASE 2

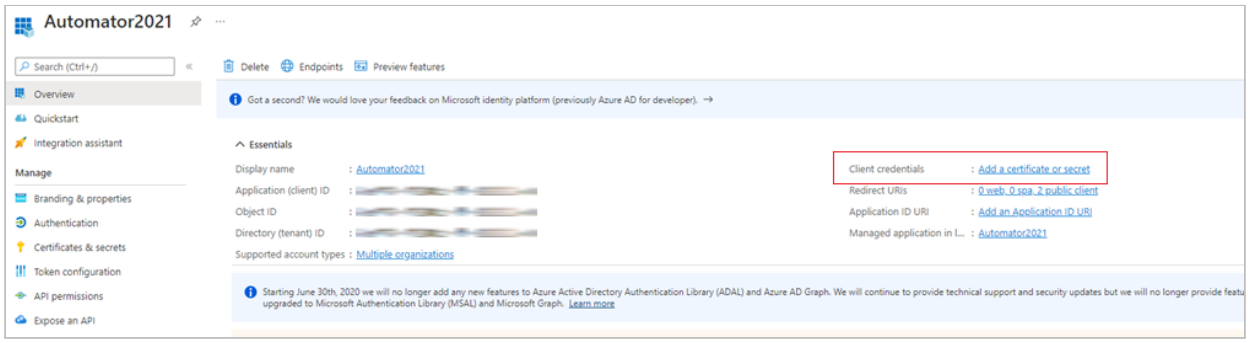
To get a Client Secret key for Power BI

1. Access Microsoft Azure to view your app registrations by clicking here.

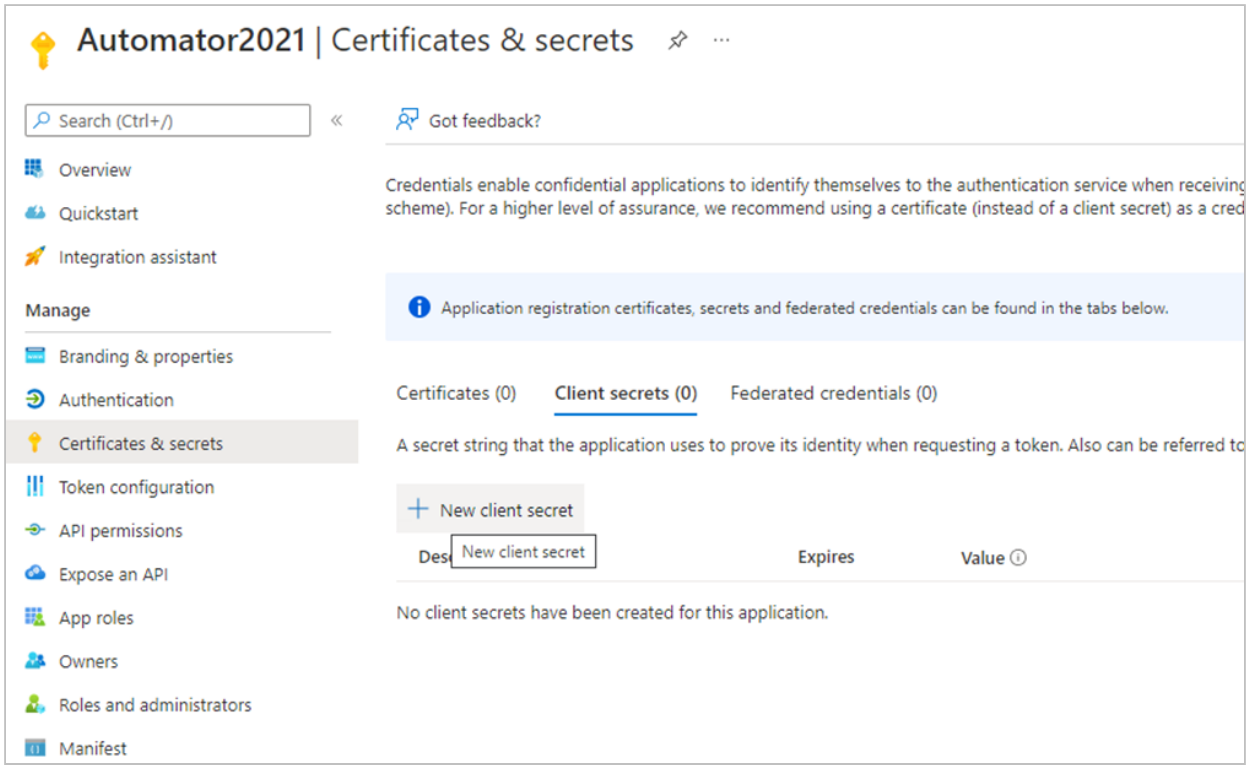
2. In the **Owned applications** tab, click the display name of your registered app (e.g., Automator2021) to view its properties, including the Application ID generated previously.



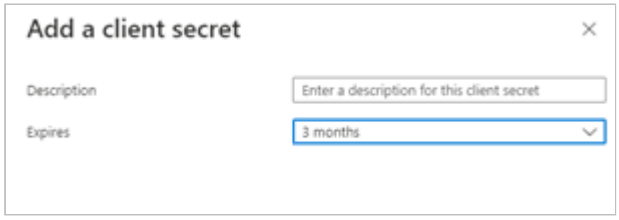
3. In the **Overview** pane of the app page, click **Add a certificate or secret**.



4. In the **Certificates & secrets** pane, open the **Client secrets** tab and create a new secret key by clicking **+New client secret**.



5. In **Add a client secret**, provide a description for the secret key, and set its expiration.



The **Client secrets** tab should now show the newly created client secret. The text combination under the **Value** column serves as the client secret key.

NOTE: The following steps should be performed to complete the integration:

a. After generating the client ID and secret key, input them in the corresponding fields in the **Server Integration** pane of Automator settings under System Settings. Then, enter the host name and port. The **Redirect URL** field should now be populated with the redirect link.

b. In Microsoft Azure, access the **Authentication** pane of the app page, and click **Add URI** to input the redirect URL generated in Monarch Server.

c. Click **Save**.

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 a DataPrep Export item in a Visual Process.
   - **DataPrep Input Files**: The named path is available to the workspace input (to replace the input path) as part of the **Replace Paths** option when a workspace is opened in the Server Library. It helps 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.
- **Google Analytics Key Files**: The named path is available to the workspace with Google Analytics data source (to replace Google Analytics Key Files in data source).
- **Monitoring**: The named path is available to monitoring path in Standard/Visual Processes.
- **Move Disposition**: The named path is available to job log disposition.
- **Project**: The named path is available to project files in Standard Processes.
- **Project Data Link Files**: The named path is available to data link files in the project (Standard Processes).
- **Project Inputs**. The named path is available to text input in the project (Standard Processes).
- **Project Models**: The named path is available to models in the project (Standard Processes).
- **Project Prerequisites.** The named path is available to prerequisites in the project (Standard Processes).
- **Publish Tableau Export Passwords**: The named path is available to the Publish to Tableau option (to select password).
- **Publish Tableau Export TabCmd**: The named path is available to the Publish to Tableau option (to select TabCmd).
- **RSS Action**: The named path is available to the RSS distribution action for Standard Processes.
- **Script References**. The named path is available to the script (to specify some references).

4. Select location in the Location field.
5. Select **Create folder allowed** to show the **Create New Folder** field that will allow the user to create a folder in the file system.
6. Select **Browse from root** to make the whole file system of a selected location is accessible to the user
   or
   Enter the path in the **Path** field. In this case only files and folder within the path selected are accessible to the user.
7. Click the **Save** button.

To edit a named path

1. Click the name of the named path you want to change or click anywhere on the row, and then click .
2. Make necessary changes. For details, see To add a new named path.

To assign rights to a named path

1. Click the **Rights** tab found beside the **Named Paths** tab.
2. If you want the selected named path to be available to all users, select the **Global** checkbox.
3. Under **User Groups**, choose the user groups to which the named path should be assigned by dragging and dropping them from the **Available User Groups** pane to the **Selected User Groups** pane.
4. Under **Roles**, choose the roles to which the named path should be assigned by dragging and dropping them from the **Available Roles** pane to the **Selected Roles** pane.

5. Under **Users**, choose the users to which the named path should be assigned by dragging and dropping them from the **Available Users** pane to the **Selected Users** pane.
6. Click the **Save** button to save all changes.

To delete a named path

1. On the **Named Paths** page select the project location you want to delete.
2. Click at the end of the row with the named path you want to delete.
3. Click **Delete** to remove the named path from the list.

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.

Select All button, then all new processes will be assigned to the exception date.

7. For the process behavior, select the **Postpone Process execution** checkbox if you want to execute the process immediately after the exception date. To let the process run on the set schedule, leave this option unchecked.
8. Set up **Exception Frequency** by either selecting a single date or indicating a daily, weekly, or monthly occurrence.

   In setting up the Exception Frequency, you can set all-day occurrence or set a specific timeframe.

NOTE: When setting a monthly occurrence, you can select the **Business Day** option. A business day is considered every official work day of the week; another common term is work(ing) day. These are the days between and including Monday through Friday.

9. Click the **Save** button.

To edit an exception date

1. Click the name of the exception date you want to change or click anywhere on the row, and then click .
2. Make necessary changes. For details, see To add a new exception date.

To delete an exception date

1. Click at the end of the row with the exception date you want to delete.
2. Click **Delete** to remove the exception date from the list.

Locale Management

A locale is a group of settings that determine the language, time format, date format, currency symbol, and so on. The existing locales are listed on the Locale Management page.

To access the Locale Management page, click **System Settings** on the Navigation Tree, and select **Locale Management**.

Use this page to edit locales.

The Locale Management page contains the following items:

- **Locale group**: Select from the list of options.
  - Select **Admin** if you need to edit labels appearing only in MSAdmin.
  - Select **Client** if you need to edit labels appearing only in MSClient.
  - Select **Common** if you need to edit labels appearing both in MSAdmin and MSClient.
  - Select **Services** if you need to edit labels appearing in System Reports.
  - Select **Database** if you need to edit labels appearing in the database.

- **Locale Language**: Select a language from the list. This list includes languages that were selected during the installation.
- **Search**: You can search for a label, by typing its name in this box.
- **Constant name**: The list displays names of fields as they appear in code, to facilitate tracking their location.
- **Original name**: The list displays all labels appearing on the user interface.
- **New Name**: The list displays all labels appearing on the user interface after localization.
- **Reload** icon : Click this icon to reload the grid.
- **Edit** icon : To edit a label, click anywhere on the row and click this icon. The **Edit Record** dialog box appears. Type the new name into a box and click **Submit**.

NOTE: You can open the **Edit Record** dialog box by clicking anywhere on a row.

You can edit labels in one of the following ways:

- One by one:
  - Click the name of a label.
  - Type a new name in the **Edit Record** dialog box.
- All at once:
  - Select the **Import Resources** tab.
  - After editing, click the **Upload** button to browse for an .xml file containing the list of all localized labels. Labels from the selected file will be uploaded and displayed on the Locale Management page.

NOTE: When necessary, company logos can be changed in Locale Management. Different logos can also be added for different locations (English, Deutsch, Français).

To do so, modify the following constants under the **Common Locale Group**.

- **Theme_LoginPageLogoUri**: To change the logo in the login page
- **Theme_HeaderLogoUri**: To change the header logo
- **Theme_ShortcutIconUri**: To change the shortcut icon

To download the edited labels, click the **Export Resources** tab.

The Export Resource page contains the following items:

- **Locale Language**: Select a language from the list. This list includes languages that were selected during the installation.

- **Export Resources**: Click this button to open or download an .xml file, containing the list of all labels.
- **Export only overridden resources**: Select this check box, to export only overridden resources.

Bulk Schedules Edits

The **Bulk Schedules Edits** feature allows you to apply bulk editing of schedules to Automator Standard Processes, Visual Processes, Pump Custodian Processes, and Maintenance Processes.

To use bulk schedule editing

1. On the **Navigation Tree**, click **System Settings**, and then select **Bulk Schedules Edits**.
2. On the displayed page, select the **Find a Schedule** tab.
3. In the **Select Process Type** field, select the process you want to edit from the drop-down list. The available options are:
   - Automator Standard Processes
   - Visual Processes
   - Pump Custodian Processes
   - 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.

System Reports

Monarch Server Administrator provides an easy means of generating system reports. The following types of system reports can be generated:

- Configuration reports provide information about the current configuration of various parts of the Monarch Server system, such as the existing roles, users, and so on.
- Transaction reports provide information about any actions that have occurred in the system, such as security, audit, and so on.

Configuration Reports

Configuration reports provide information about the current configuration of various parts of the Monarch Server system, such as the existing roles, users, and so on. You can generate the following types of configuration reports:

- Product License: Displays the product licensee, the product expiration date (if any), etc.
- Roles: Lists the existing roles and their assigned user groups and users.
- User Groups: Lists the existing user groups, their assigned users, and their privileges.
- Users: Lists the existing users, their e-mail addresses, and their locked privileges.
- 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 view configuration reports

1. On the Navigation Tree, select **System Reports** and then select **Configuration**. The **Configuration Reports** page displays.
2. On the Configuration Reports page, select the link that corresponds to the desired report type. Note that all configuration reports can be printed or exported to the following formats: *.pdf, *.csv, *.xls, *.rtf, *.tiff, and *.mhtml.

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 (e.g., Report View, Chart, and so on)
- Product options (e.g., maximum allowed connections, machine names, and so on)

To view, export, and print a Product License configuration report

1. On the Navigation Tree, select **System Reports** and then select **Configuration**.
2. Select the **Product License** link. The Product License configuration report page displays.
3. Choose between the following views:
    - **Grid**: The Grid view displays the report in tabular format. For each column, select any of the following operations by clicking ⋮:
        - **Sort Ascending**: Select this option to arrange the entries in ascending order.
        - **Sort Descending**: Select this option to arrange the entries in descending order.
        - **Columns**: Select the columns to display in the grid view.
        - **Set Column Position**: Choose between Move Previous or Move Next to arrange the columns.

      The following options are also available in Grid view:
        - **Export to Excel**: Select this option to export the report as an Excel Worksheet (*.xlsx).
        - **Export to PDF**: Select this option to export the report as a PDF file (*.pdf).
        - **Search**: Enter a keyword to perform a search within the entries.
        - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
        - ***X* items per page**: Choose the number of items to display from the dropdown menu.
        - **Refresh**: Click the **Refresh** button to display the most recent reports.
    - **Report**: The Report view displays the report in textual format. The following options are available:
        - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
        - **Export to the selected format**: Choose from among the following export format types that are listed in the dropdown menu: Acrobat (PDF) file, CSV (comma delimited), Excel 97-2003, Rich Text Format, TIFF file, and Web Archive. Click the **Export** button to complete the export process.
        - **Refresh**: Click the **Refresh** button to display the most recent reports.

- **Print Preview**: Click the **Print Preview** button to view the report in printed format.
- **Print**: Click the **Print** button at the top of the page, then click **OK** in the **Print** dialog.

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, export, 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 page is displayed.
3. Choose between the following views:
   - **Grid**: The Grid view displays the report in tabular format. For each column, select any of the following operations by clicking :
     - **Sort Ascending**: Select this option to arrange the entries in ascending order.
     - **Sort Descending**: Select this option to arrange the entries in descending order.
     - **Columns**: Select the columns to display in the grid view.
     - **Filter**: Define the settings to filter the entries shown.
     - **Group column**: Combine identical entries into a single group, with each unique entry displayed once. Remove the grouping by clicking next to the name of the grouped column or by clicking again and then selecting **Ungroup column**.
     - **Set Column Position**: Choose between **Move Previous** or **Move Next** to arrange the columns.

     The following options are also available in Grid view:
     - **Export to Excel**: Select this option to export the report as an Excel Worksheet (*.xlsx).
     - **Export to PDF**: Select this option to export the report as a PDF file (*.pdf).

        - **Search**: Enter a keyword to perform a search within the entries.
        - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
        - ***X* items per page**: Choose the number of items to display from the dropdown menu.
        - **Refresh**: Click the **Refresh** button to display the most recent reports.
    - **Report**: The Report view displays the report in textual format. The following options are available:
        - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
        - **Export to the selected format**: Choose from among the following export format types that are listed in the dropdown menu: Acrobat (PDF) file, CSV (comma delimited), Excel 97-2003, Rich Text Format, TIFF file, and Web Archive. Click the **Export** button to complete the export process.
        - **Refresh**: Click the **Refresh** button to display the most recent reports.
        - **Print Preview**: Click the **Print Preview** button to view the report in printed format.
        - **Print**: Click the **Print** button at the top of the page, then click **OK** in the **Print** dialog.

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, export, 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 page is displayed.
3. Choose between the following views:
    - **Grid**: The Grid view displays the report in tabular format. For each column, select any of the following operations by clicking :
        - **Sort Ascending**: Select this option to arrange the entries in ascending order.
        - **Sort Descending**: Select this option to arrange the entries in descending order.
        - **Columns**: Select the columns to display in the grid view.

- **Filter**: Define the settings to filter the entries shown.
- **Group column**: Combine identical entries into a single group, with each unique entry displayed once. Remove the grouping by clicking next to the name of the grouped column or by clicking again and then selecting **Ungroup column**.
- **Set Column Position**: Choose between **Move Previous** or **Move Next** to arrange the columns.

The following options are also available in Grid view:

- **Export to Excel**: Select this option to export the report as an Excel Worksheet (*.xlsx).
- **Export to PDF**: Select this option to export the report as a PDF file (*.pdf).
- **Search**: Enter a keyword to perform a search within the entries.
- **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
- ***X* items per page**: Choose the number of items to display from the dropdown menu.
- **Refresh**: Click the **Refresh** button to display the most recent reports.

◦ **Report**: The Report view displays the report in textual format. The following options are available:

- **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
- **Export to the selected format**: Choose from among the following export format types that are listed in the dropdown menu: Acrobat (PDF) file, CSV (comma delimited), Excel 97-2003, Rich Text Format, TIFF file, and Web Archive. Click the **Export** button to complete the export process.
- **Refresh**: Click the **Refresh** button to display the most recent reports.
- **Print Preview**: Click the **Print Preview** button to view the report in printed format.
- **Print**: Click the **Print** button at the top of the page, then click **OK** in the **Print** dialog.

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 page is displayed.
3. Choose between the following views:
    - **Grid**: The Grid view displays the report in tabular format. For each column, select any of the following operations by clicking :
        - **Sort Ascending**: Select this option to arrange the entries in ascending order.
        - **Sort Descending**: Select this option to arrange the entries in descending order.
        - **Columns**: Select the columns to display in the grid view.
        - **Filter**: Define the settings to filter the entries shown.
        - **Group column**: Combine identical entries into a single group, with each unique entry displayed once. Remove the grouping by clicking next to the name of the grouped column or by clicking again and then selecting **Ungroup column**.
        - **Set Column Position**: Choose between **Move Previous** or **Move Next** to arrange the columns.

      The following options are also available in Grid view:
        - **Export to Excel**: Select this option to export the report as an Excel Worksheet (*.xlsx).
        - **Export to PDF**: Select this option to export the report as a PDF file (*.pdf).
        - **Search**: Enter a keyword to perform a search within the entries.
        - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
        - ***X* items per page**: Choose the number of items to display from the dropdown menu.
        - **Refresh**: Click the **Refresh** button to display the most recent reports.
    - **Report**: The Report view displays the report in textual format. The following options are available:
        - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
        - **Export to the selected format**: Choose from among the following export format types that are listed in the dropdown menu: Acrobat (PDF) file, CSV (comma delimited), Excel 97-2003, Rich Text Format, TIFF file, and Web Archive. Click the **Export** button to complete the export process.
        - **Refresh**: Click the **Refresh** button to display the most recent reports.
        - **Print Preview**: Click the **Print Preview** button to view the report in printed format.
        - **Print**: Click the **Print** button at the top of the page, then click **OK** in the **Print** dialog.

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, export, 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 page is displayed.
3. Choose between the following views:
    - **Grid**: The Grid view displays the report in tabular format. For each column, select any of the following operations by clicking ⋮:
        - **Sort Ascending**: Select this option to arrange the entries in ascending order.
        - **Sort Descending**: Select this option to arrange the entries in descending order.
        - **Columns**: Select the columns to display in the grid view.
        - **Filter**: Define the settings to filter the entries shown.
        - **Group column**: Combine identical entries into a single group, with each unique entry displayed once. Remove the grouping by clicking ⊗ next to the name of the grouped column or by clicking ⋮ again and then selecting **Ungroup column**.
        - **Set Column Position**: Choose between **Move Previous** or **Move Next** to arrange the columns.

      The following options are also available in Grid view:

        - **Export to Excel**: Select this option to export the report as an Excel Worksheet (*.xlsx).
        - **Export to PDF**: Select this option to export the report as a PDF file (*.pdf).
        - **Search**: Enter a keyword to perform a search within the entries.
        - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
        - ***X* items per page**: Choose the number of items to display from the dropdown menu.
        - **Refresh**: Click the **Refresh** button to display the most recent reports.

- **Report**: The Report view displays the report in textual format. The following options are available:
  - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
  - **Export to the selected format**: Choose from among the following export format types that are listed in the dropdown menu: Acrobat (PDF) file, CSV (comma delimited), Excel 97-2003, Rich Text Format, TIFF file, and Web Archive. Click the **Export** button to complete the export process.
  - **Refresh**: Click the **Refresh** button to display the most recent reports.
  - **Print Preview**: Click the **Print Preview** button to view the report in printed format.
  - **Print**: Click the **Print** button at the top of the page, then click **OK** in the **Print** dialog.

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

To view, export, 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. Choose between the following views:
   - **Grid**: The Grid view displays the report in tabular format. For each column, select any of the following operations by clicking :
     - **Sort Ascending**: Select this option to arrange the entries in ascending order.
     - **Sort Descending**: Select this option to arrange the entries in descending order.
     - **Columns**: Select the columns to display in the grid view.

- **Filter**: Define the settings to filter the entries shown.
- **Group column**: Combine identical entries into a single group, with each unique entry displayed once. Remove the grouping by clicking next to the name of the grouped column or by clicking again and then selecting **Ungroup column**.
- **Set Column Position**: Choose between **Move Previous** or **Move Next** to arrange the columns.

The following options are also available in Grid view:

- **Export to Excel**: Select this option to export the report as an Excel Worksheet (*.xlsx).
- **Export to PDF**: Select this option to export the report as a PDF file (*.pdf).
- **Search**: Enter a keyword to perform a search within the entries.
- **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
- ***X* items per page**: Choose the number of items to display from the dropdown menu.
- **Refresh**: Click the **Refresh** button to display the most recent reports.

◦ **Report**: The Report view displays the report in textual format. The following options are available:

- **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
- **Export to the selected format**: Choose from among the following export format types that are listed in the dropdown menu: Acrobat (PDF) file, CSV (comma delimited), Excel 97-2003, Rich Text Format, TIFF file, and Web Archive. Click the **Export** button to complete the export process.
- **Refresh**: Click the **Refresh** button to display the most recent reports.
- **Print Preview**: Click the **Print Preview** button to view the report in printed format.
- **Print**: Click the **Print** button at the top of the page, then click **OK** in the **Print** dialog.

NOTE: For information on how to convert a report to another format, see Converting a report to another format.

Transaction Reports

Transaction reports provide information about any actions that have occurred in the system, such as security, audit, and so on.

You can generate the following types of transaction reports:

- Security — Lists the time, user name, and log on/log off information.
- Added Users — Lists information about users add during the date range selected.

- Deleted/Suspended Users — Lists information about users who have been deleted or suspended.
- Users Not Logged In — Lists information users who have not logged in over a date range.
- Audit Entity Actions — Lists information about audit entity actions.
- Custodian Schedules Logs — Lists information about custodian schedules events.
- Automator Distribution Audit – Lists all the export and distribution (move and copy) paths used in processes and all email distributions used in distribution actions for a specified period.

To view transaction reports

1. On the **Navigation Tree**, click **System Reports**, and select **Transaction**. The Transaction Reports page appears. The Transaction Reports page contains the following items:
    - **Events From To**: Specify the beginning and the end of the time interval, for which you want to remove events.
    - **Clear Log**: Click this button to remove all events that occurred during the time interval defined earlier.
    - **Transaction Reports**: Contains links to transaction reports of different types.
2. To generate a report, on the Transaction Reports page, click the link that corresponds to the desired report type.

Security Report

A Security Report is one of transaction reports provided by Monarch Server. This report provides the following information:

- Date and time the user(s) log on to and log off from the system
- Name(s) of the user(s) who logged on to and log off from the system
- Date and time the user(s) log on to the system with incorrect password
- Name(s) of the user(s) who logged on to the system with incorrect password

To view and print a Security transaction report

1. On the **Navigation Tree**, click **System Reports** and select **Transaction**.
2. Click the **Security** link. The Security Transaction Report page is displayed.
3. In the **From ... To** fields specify a time period, for which you want to view the security actions. Click to open the Calendar window, and select the date.

> **NOTE**: If you do not provide a date or time period, all actions for the specified user(s) will be displayed.

4. From the **User** list, select a user, whose actions you want to include in the report. (To view security information for all users, select **All Users**.)
5. Click **View** to display the report in either tabular (Grid) or textual (Report) format.

- **Grid:** The Grid view displays the report in tabular format. For each column, select any of the following operations by clicking :
  - **Sort Ascending**: Select this option to arrange the entries in ascending order.
  - **Sort Descending**: Select this option to arrange the entries in descending order.
  - **Columns**: Select the columns to display in the grid view.
  - **Filter**: Define the settings to filter the entries shown.
  - **Group column**: Combine identical entries into a single group, with each unique entry displayed once. Remove the grouping by clicking next to the name of the grouped column or by clicking again and then selecting **Ungroup column**.
  - **Set Column Position**: Choose between **Move previous** or **Move next** to arrange the columns.

  The following options are also available in Grid view:
  - **Export to Excel**: Select this option to export the report as an Excel Worksheet (*.xlsx).
  - **Export to PDF**: Select this option to export the report as a PDF file (*.pdf).
  - **Search**: Enter a keyword to perform a search within the entries.
  - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
  - ***X* items per page**: Choose the number of items to display from the dropdown menu.
  - **Refresh**: Click the **Refresh** button to display the most recent reports.
- **Report**: The Report view displays the report in textual format. The following options are available:
  - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
  - **Export to the selected format**: Choose from among the following export format types that are listed in the dropdown menu: Acrobat (PDF) file, CSV (comma delimited), Excel 97-2003, Rich Text Format, TIFF file, and Web Archive. Click the **Export** button to complete the export process.
  - **Refresh**: Click the **Refresh** button to display the most recent reports.
  - **Print preview**: Click the **Print Preview** button to view the report in printed format.
  - **Print**: Click the **Print** button at the top of the page, then click **OK** in the **Print** dialog.

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 that the maximum number of days is 90.
4. Click **View** to display the report in either tabular (Grid) or textual (Report) format.
   - **Grid:** The Grid view displays the report in tabular format. For each column, select any of the following operations by clicking :
     - **Sort Ascending**: Select this option to arrange the entries in ascending order.
     - **Sort Descending**: Select this option to arrange the entries in descending order.
     - **Columns**: Select the columns to display in the grid view.
     - **Filter**: Define the settings to filter the entries shown.
     - **Group column**: Combine identical entries into a single group, with each unique entry displayed once. Remove the grouping by clicking next to the name of the grouped column or by clicking again and then selecting **Ungroup column**.
     - **Set Column Position**: Choose between **Move previous** or **Move next** to arrange the columns.

     The following options are also available in Grid view:
     - **Export to Excel**: Select this option to export the report as an Excel Worksheet (*.xlsx).
     - **Export to PDF**: Select this option to export the report as a PDF file (*.pdf).
     - **Search**: Enter a keyword to perform a search within the entries.
     - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
     - ***X* items per page**: Choose the number of items to display from the dropdown menu.
     - **Refresh**: Click the **Refresh** button to display the most recent reports.
   - **Report**: The Report view displays the report in textual format. The following options are available:
     - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.

- **Export to the selected format**: Choose from among the following export format types that are listed in the dropdown menu: Acrobat (PDF) file, CSV (comma delimited), Excel 97-2003, Rich Text Format, TIFF file, and Web Archive. Click the **Export** button to complete the export process.
- **Refresh**: Click the **Refresh** button to display the most recent reports.
- **Print preview**: Click the **Print Preview** button to view the report in printed format.
- **Print**: Click the **Print** button at the top of the page, then click **OK** in the **Print** dialog.

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** to display the report in either tabular (Grid) or textual (Report) format.
   - **Grid:** The Grid view displays the report in tabular format. For each column, select any of the following operations by clicking :
     - **Sort Ascending**: Select this option to arrange the entries in ascending order.
     - **Sort Descending**: Select this option to arrange the entries in descending order.
     - **Columns**: Select the columns to display in the grid view.
     - **Filter**: Define the settings to filter the entries shown.
     - **Group column**: Combine identical entries into a single group, with each unique entry displayed once. Remove the grouping by clicking next to the name of the grouped column or by clicking again and then selecting **Ungroup column**.

- **Set Column Position**: Choose between **Move previous** or **Move next** to arrange the columns.

The following options are also available in Grid view:

- **Export to Excel**: Select this option to export the report as an Excel Worksheet (*.xlsx).
- **Export to PDF**: Select this option to export the report as a PDF file (*.pdf).
- **Search**: Enter a keyword to perform a search within the entries.
- **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
- ***X* items per page**: Choose the number of items to display from the dropdown menu.
- **Refresh**: Click the **Refresh** button to display the most recent reports.

◦ **Report**: The Report view displays the report in textual format. The following options are available:

- **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
- **Export to the selected format**: Choose from among the following export format types that are listed in the dropdown menu: Acrobat (PDF) file, CSV (comma delimited), Excel 97-2003, Rich Text Format, TIFF file, and Web Archive. Click the **Export** button to complete the export process.
- **Refresh**: Click the **Refresh** button to display the most recent reports.
- **Print preview**: Click the **Print Preview** button to view the report in printed format.
- **Print**: Click the **Print** button at the top of the page, then click **OK** in the **Print** dialog.

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** to display the report in either tabular (Grid) or textual (Report) format.

   - **Grid:** The Grid view displays the report in tabular format. For each column, select any of the following operations by clicking :

     - **Sort Ascending**: Select this option to arrange the entries in ascending order.
     - **Sort Descending**: Select this option to arrange the entries in descending order.
     - **Columns**: Select the columns to display in the grid view.
     - **Filter**: Define the settings to filter the entries shown.
     - **Group column**: Combine identical entries into a single group, with each unique entry displayed once. Remove the grouping by clicking next to the name of the grouped column or by clicking again and then selecting **Ungroup column**.
     - **Set Column Position**: Choose between **Move previous** or **Move next** to arrange the columns.

     The following options are also available in Grid view:

     - **Export to Excel**: Select this option to export the report as an Excel Worksheet (*.xlsx).
     - **Export to PDF**: Select this option to export the report as a PDF file (*.pdf).
     - **Search**: Enter a keyword to perform a search within the entries.
     - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
     - ***X* items per page**: Choose the number of items to display from the dropdown menu.
     - **Refresh**: Click the **Refresh** button to display the most recent reports.

   - **Report**: The Report view displays the report in textual format. The following options are available:

     - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
     - **Export to the selected format**: Choose from among the following export format types that are listed in the dropdown menu: Acrobat (PDF) file, CSV (comma delimited), Excel 97-2003, Rich Text Format, TIFF file, and Web Archive. Click the **Export** button to complete the export process.
     - **Refresh**: Click the **Refresh** button to display the most recent reports.
     - **Print preview**: Click the **Print Preview** button to view the report in printed format.
     - **Print**: Click the **Print** button at the top of the page, then click **OK** in the **Print** dialog.

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** to display the report in either tabular (Grid) or textual (Report) format.
   - **Grid:** The Grid view displays the report in tabular format. For each column, select any of the following operations by clicking :
     - **Sort Ascending**: Select this option to arrange the entries in ascending order.
     - **Sort Descending**: Select this option to arrange the entries in descending order.
     - **Columns**: Select the columns to display in the grid view.
     - **Filter**: Define the settings to filter the entries shown.
     - **Group column**: Combine identical entries into a single group, with each unique entry displayed once. Remove the grouping by clicking next to the name of the grouped column or by clicking again and then selecting **Ungroup column**.
     - **Set Column Position**: Choose between **Move previous** or **Move next** to arrange the columns.

     The following options are also available in Grid view:

     - **Export to Excel**: Select this option to export the report as an Excel Worksheet (*.xlsx).

        - **Export to PDF**: Select this option to export the report as a PDF file (*.pdf).
        - **Search**: Enter a keyword to perform a search within the entries.
        - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
        - ***X* items per page**: Choose the number of items to display from the dropdown menu.
        - **Refresh**: Click the **Refresh** button to display the most recent reports.
    - **Report**: The Report view displays the report in textual format. The following options are available:
        - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
        - **Export to the selected format**: Choose from among the following export format types that are listed in the dropdown menu: Acrobat (PDF) file, CSV (comma delimited), Excel 97-2003, Rich Text Format, TIFF file, and Web Archive. Click the **Export** button to complete the export process.
        - **Refresh**: Click the **Refresh** button to display the most recent reports.
        - **Print preview**: Click the **Print Preview** button to view the report in printed format.
        - **Print**: Click the **Print** button at the top of the page, then click **OK** in the **Print** dialog.

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** to display the report in either tabular (Grid) or textual (Report) format.
   - **Grid:** The Grid view displays the report in tabular format. For each column, select any of the following operations by clicking :
     - **Sort Ascending**: Select this option to arrange the entries in ascending order.
     - **Sort Descending**: Select this option to arrange the entries in descending order.
     - **Columns**: Select the columns to display in the grid view.
     - **Filter**: Define the settings to filter the entries shown.
     - **Group column**: Combine identical entries into a single group, with each unique entry displayed once. Remove the grouping by clicking next to the name of the grouped column or by clicking again and then selecting **Ungroup column**.
     - **Set Column Position**: Choose between **Move previous** or **Move next** to arrange the columns.

     The following options are also available in Grid view:
     - **Export to Excel**: Select this option to export the report as an Excel Worksheet (*.xlsx).
     - **Export to PDF**: Select this option to export the report as a PDF file (*.pdf).
     - **Search**: Enter a keyword to perform a search within the entries.
     - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
     - ***X* items per page**: Choose the number of items to display from the dropdown menu.
     - **Refresh**: Click the **Refresh** button to display the most recent reports.
   - **Report**: The Report view displays the report in textual format. The following options are available:
     - **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
     - **Export to the selected format**: Choose from among the following export format types that are listed in the dropdown menu: Acrobat (PDF) file, CSV (comma delimited), Excel 97-2003, Rich Text Format, TIFF file, and Web Archive. Click the **Export** button to complete the export process.
     - **Refresh**: Click the **Refresh** button to display the most recent reports.
     - **Print preview**: Click the **Print Preview** button to view the report in printed format.
     - **Print**: Click the **Print** button at the top of the page, then click **OK** in the **Print** dialog.

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** to display the report in either tabular (Grid) or textual (Report) format.
    - **Grid:** The Grid view displays the report in tabular format. For each column, select any of the following operations by clicking ⋮:
        - **Sort Ascending**: Select this option to arrange the entries in ascending order.
        - **Sort Descending**: Select this option to arrange the entries in descending order.
        - **Columns**: Select the columns to display in the grid view.
        - **Filter**: Define the settings to filter the entries shown.

- **Group column**: Combine identical entries into a single group, with each unique entry displayed once. Remove the grouping by clicking next to the name of the grouped column or by clicking again and then selecting **Ungroup column**.
- **Set Column Position**: Choose between **Move previous** or **Move next** to arrange the columns.

The following options are also available in Grid view:

- **Export to Excel**: Select this option to export the report as an Excel Worksheet (*.xlsx).
- **Export to PDF**: Select this option to export the report as a PDF file (*.pdf).
- **Search**: Enter a keyword to perform a search within the entries.
- **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
- ***X* items per page**: Choose the number of items to display from the dropdown menu.
- **Refresh**: Click the **Refresh** button to display the most recent reports.

◦ **Report**: The Report view displays the report in textual format. The following options are available:

- **Page navigation**: Enter a page number or click the navigation arrows to switch pages.
- **Export to the selected format**: Choose from among the following export format types that are listed in the dropdown menu: Acrobat (PDF) file, CSV (comma delimited), Excel 97-2003, Rich Text Format, TIFF file, and Web Archive. Click the **Export** button to complete the export process.
- **Refresh**: Click the **Refresh** button to display the most recent reports.
- **Print preview**: Click the **Print Preview** button to view the report in printed format.
- **Print**: Click the **Print** button at the top of the page, then click **OK** in the **Print** dialog.

NOTE: For information on how to convert a report to another format, see Converting a report to another format.

Converting a report to another format

To convert a report to another format

Select required format from the **Export to the selected format** list at the top of the page and click **Export**.
Formats available:

◦ Acrobat (PDF) file

◦ CSV (comma delimited)

◦ Excel

- Rich Text Format
- TIFF file
- Web Archive

Active Users

Monarch Server provides the possibility to monitor active users in the system.

To access the Active Users Report, on the **Navigation Tree**, click **System Reports**, and select **Active Users**. The list displays the following information:

- User name
- Privileges
- Creation time
- Module name
- Terminate

Click the icon in the **Terminate** column to close the current session for the selected active user. The **Terminate User Session** dialog box will pop up with the the following question: *Terminate selected User Session?*

Click **Delete** to confirm your choice or **Cancel** to remain the session.

Technical Support

Software Support Policy

For details about the latest product versions, product development, software releases, and maintenance and support policy, reach out to Altair Support.

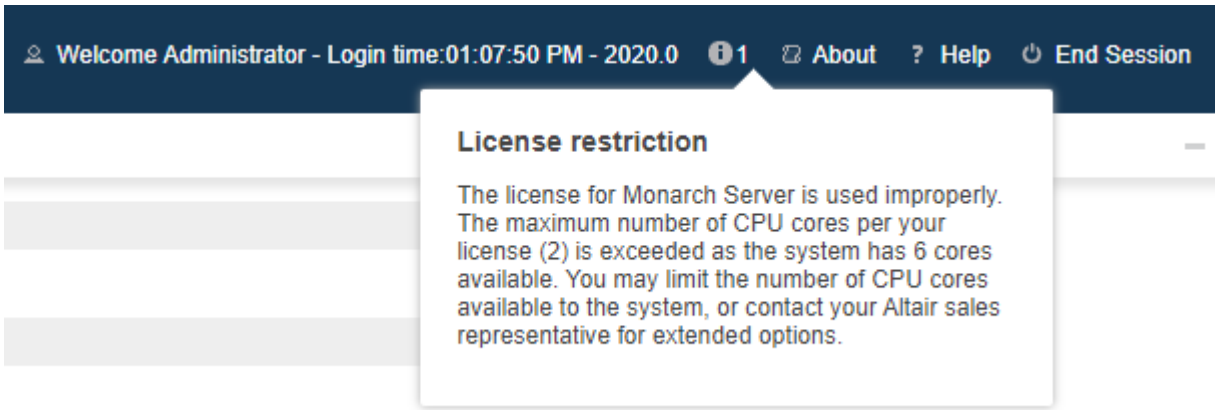
Before Contacting Technical Support

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.
- Reach out to Altair Support.

> **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

Portal: https://altair.com/customer-support

© Altair Engineering, Inc. All Rights Reserved.