



Qlik Enterprise Manager Setup and User Guide

Qlik Enterprise Manager™

May 2025

Last updated: November 12, 2025

Copyright © 1993-2025 QlikTech International AB. All rights reserved.

1 What's new?	9
1.1 Clone Task API	9
2 Introduction	10
2.1 Example Enterprise Manager environment	11
2.2 Enterprise Manager architecture	12
3 Terminology	13
3.1 Change Data Capture (CDC)	13
3.2 Full load	13
3.3 Apply latency	13
Latency when applying large transactions	13
Latency when no transactions are being applied	13
3.4 Source latency	13
3.5 Target latency	13
3.6 Overall latency	14
3.7 Source endpoint	14
3.8 Target endpoint	14
3.9 Net Changes table	14
4 Installation and setup	15
4.1 Preparing your system for Enterprise Manager	15
Hardware configuration guidelines	15
Sizing guidelines	16
Software requirements	16
Compatibility with related Qlik Products	17
Replication Management license	17
4.2 Installing Enterprise Manager	18
Installing Qlik Enterprise Manager using the Setup Wizard	19
Upgrading Enterprise Manager	19
Migrating the Enterprise Manager Repository	20
Silently installing Enterprise Manager	21
Silently upgrading Enterprise Manager	22
Silently uninstalling Enterprise Manager	23
4.3 Changing the Enterprise Manager service account	24
4.4 Setting the login authentication method	25
Setting Single Sign-on authentication	25
Setting Single Sign-on authentication with Kerberos	25
Setting form authentication	26
Setting up SAML authentication	27
Setting up Personal Access Token authentication for the API	30
Logging in to Enterprise Manager with a JSON Web Token via the API	35
4.5 Starting to work with the Qlik Enterprise Manager Console	37
Registering Licenses	38
5 Security considerations	39
5.1 Setting up HTTPS for the Enterprise Manager console	39
Checking if an SSL Certificate is installed	39
Using the self-signed certificate	39
5.2 Setting up HSTS on Enterprise Manager	42

Enabling HSTS	42
Disabling HSTS	42
5.3 Replacing the self-signed certificate on Windows	43
5.4 Setting the hostname and changing the SSL port	44
5.5 Replacing the Master User Password	45
The Master Key	45
High Availability mode	46
5.6 Encrypting the user permissions file	46
5.7 Controlling execution of user-defined commands	47
Executing operating system commands as a different user	48
6 Managing servers	49
6.1 Server requirements	49
Qlik Replicate Server requirements	49
Qlik Compose Server requirements	50
6.2 Adding Servers	50
6.3 Monitoring servers	52
Customizing server columns	53
Searching for servers	53
6.4 Server settings	54
Global error handling	54
Resource Control	55
File Transfer Service	56
External utilities	58
Logging	59
More options	62
Server management permissions	62
6.5 Additional server management options	64
6.6 Registering a license for a monitored server	66
6.7 Viewing server messages	68
7 Defining and managing tasks	69
7.1 Adding tasks	69
Bidirectional replication	71
7.2 Editing and viewing a task description	74
7.3 Adding a source and target endpoint to a task	74
7.4 Selecting tables and/or views for replication	76
Searching for tables/views to use in a replication task	77
Selecting specific tables/views for replication	78
Setting load order	79
Removing specific tables/views from a replication task	80
Creating table/view selection patterns	80
7.5 Editing a replication task	82
7.6 Searching for tasks	82
7.7 Deleting a replication task	83
7.8 Exporting and importing tasks	83
8 Defining and managing endpoints	85
8.1 Defining an endpoint	85

8.2	Setting user permissions for a specific endpoint	86
8.3	Editing endpoint configuration information	88
8.4	Viewing endpoint configuration information	89
8.5	Testing an endpoint connection	89
8.6	Duplicating endpoints	90
8.7	Searching for endpoints	90
8.8	Deleting endpoints	90
9	Customizing tasks	91
9.1	Table Settings	91
	Performing General tasks for a single table/view	92
	Defining transformations for a single table/view	93
	Using filters	103
	Parallel Load	110
	Handling LOB columns	114
	Message format	118
	Full Load	119
9.2	Defining global rules	119
	Starting the Global Transformation Rules wizard	120
	Starting the Global Filter Rules wizard	130
	Managing global rules	130
	Starting the Global Filter Rules wizard	144
	Managing global rules	148
9.3	Using the Expression Builder	149
	Overview of the Expression Builder	149
	Build an expression	151
	Parse an expression	152
	Test an expression	152
	Using elements in the Expression Builder	154
9.4	Task Settings	179
	Metadata	180
	Bidirectional	186
	Full Load	187
	Change processing	190
	Error handling	203
	Logging	210
	Character substitution	211
	File uploads	213
	Message format	214
	Transformations and filters	219
	More options	220
10	Monitoring and controlling tasks	221
10.1	Monitoring Replicate tasks	221
	Task progress summary	221
	Viewing specific tasks	225
	Monitoring Full Load replication	228
	Monitoring Change Processing replication	233
10.2	Monitoring Compose tasks and workflows	240

Task progress summary	240
Monitoring Data Lake tasks	243
Monitoring Data Warehouse tasks	244
Monitoring workflows	245
10.3 Searching for tasks	248
10.4 Customizing task columns	249
10.5 Grouping tasks	250
10.6 Running a task	252
How to run a task	253
Using the Run button options	253
Start Processing	254
Reload target	254
Advanced run options	254
Recovering from data folder loss or corruption	261
10.7 Error handling	262
Task error handling	262
Data error handling	263
10.8 Using the monitor tools	263
Logging	263
Downloading a memory report	265
Downloading a diagnostics package	266
10.9 Scheduling jobs	266
11 Messages and notifications	269
11.1 Message Center overview	269
11.2 Customizing the Message Center	271
Searching for messages	272
11.3 Viewing additional information	272
11.4 Notifications	272
Setting a task notification	273
Setting a server notification	280
Managing notifications	284
Required permissions	284
Event IDs in Windows Event Log	285
11.5 Viewing and downloading log files	286
12 Administration	289
12.1 Enterprise Manager settings	289
Enterprise Manager logging settings	289
Message Center purge settings	291
Repository connection settings	292
Qlik Catalog Server connection	293
Analytics - Data collection and purge settings	293
Configuring mail server settings	296
Registering and managing licenses	296
12.2 User permissions	298
Encrypting the User Permissions File	299
Granular access control	300

Roles and permissions	304
Working with local groups	306
Managing user permissions	307
Managing Personal Access Tokens	309
12.3 Creating an audit trail	309
Decoding an encoded stream payload	311
13 Configuring Enterprise Manager using the CLI	313
13.1 Setting the Enterprise Manager host address	313
13.2 Setting the Enterprise Manager HTTP port	313
13.3 Setting the Enterprise Manager HTTPS port	314
13.4 Setting the Enterprise Manager root URL	314
13.5 Showing the Enterprise Manager version	314
13.6 Showing the Enterprise Manager CLI Help	314
13.7 Service Commands	314
13.8 Cleaning the self-signed certificate	315
13.9 Setting the audit trail retention size and age	315
13.10 Master User Password commands	315
Generating a random Master User Password	315
Setting or changing the MUK (Master User Key)	315
Setting or changing the Java MUK (Master User Key)	316
13.11 Showing the connectivity and login settings	317
Connectivity settings	317
SAML settings	317
13.12 Fine tuning performance	317
Turning off the Analytics Server	318
Changing the update intervals	318
14 Cataloging tasks in Qlik Catalog	320
14.1 Terminology	320
14.2 Prerequisites	321
14.3 Setting up connectivity to Qlik Catalog	321
Catalog columns	321
14.4 Limitations and considerations	321
14.5 Catalog operations	322
Cataloging tasks	322
Uncataloging tasks	324
Recataloging tasks	324
15 Analytics	325
15.1 Prerequisites	325
Install PostgreSQL	325
Create a dedicated database and assign the required privileges	326
Configure connectivity to PostgreSQL	326
Set up data collection and purging from PostgreSQL	326
Register a license	326
Obtaining a license	326
Port	327
Hardware	327

15.2 Permissions	327
15.3 Analytics dashboards	327
Trends	328
Trends by server	329
Trends by tasks	330
Top servers	331
Top tasks	332
Capacity planning	333
15.4 Exporting to TSV	335
15.5 Creating filters	335
15.6 Using the Pivot Menu	337
15.7 Analytics repository schema	337
aem_endpoint_type	337
aem_meta_source_database	338
aem_meta_target_database	338
aem_server	338
aem_source_database	338
aem_target_database	339
aem_target_processes	339
aem_task	339
aem_task_name	342
aem_task_previous_metrics	343
aem_task_profile	343
aem_task_state	343
aem_task_stop_reason	343
Sample Queries	343
A Setting up High Availability	347
A.1 Installing Qlik Enterprise Manager in a Windows cluster	347
A.2 Upgrading Qlik Enterprise Manager in a Windows cluster	350
A.3 Uninstalling Qlik Enterprise Manager from a Windows cluster	352
B Impact of DST change on Qlik Replicate	355

1 What's new?

This section describes the new and enhanced features in Enterprise Manager May 2025.



In addition to these release notes, customers who are not upgrading from the latest GA version are advised to review the release notes for all versions released since their current version.

Customers should also review the Enterprise Manager release notes in [Qlik Community](#) for information about the following:

- Migration and upgrade
- End of life/support features
- Newly supported OS platforms, databases, and third-party software versions.
- Resolved issues
- Known issues

1.1 Clone Task API

The new Clone Task API allows customers to programmatically clone existing tasks and optionally specify new names for various settings. This is especially useful for organizations that need to create numerous tasks with similar settings.

For more information, refer to the [Enterprise Manager API Guide](#).

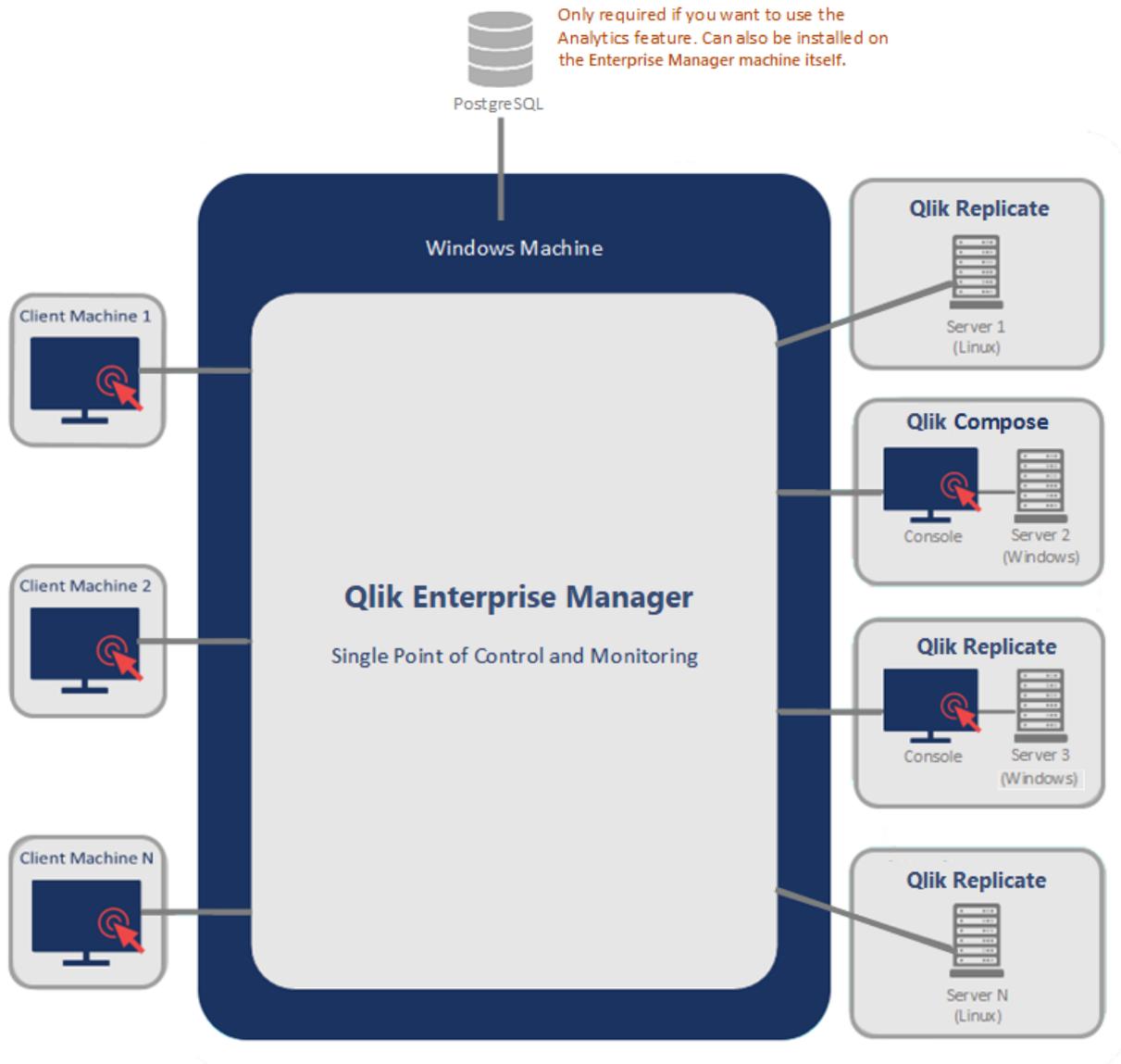
2 Introduction

Qlik Enterprise Manager, also referred to as Enterprise Manager, provides a single point of control for designing, executing, and monitoring Qlik Replicate and Compose tasks throughout your organization. If your site has multiple Qlik servers with tens, if not hundreds of tasks, Enterprise Manager greatly eases the design, management, and monitoring of these tasks. Whether your site deploys a single Qlik server or multiple servers, Enterprise Manager is your single go-to interface to create data endpoints, design tasks, execute them, and monitor the replication process in near real-time. In addition, Enterprise Manager lets you view all tasks in a tabular format that offers advanced grouping and filtering capabilities.

The following figures show a high-level view of a possible Enterprise Manager installation environment and a more detailed architecture diagram.

Note that components labeled as "Qlik Server" can either be Qlik Replicate or Qlik Compose.

2.1 Example Enterprise Manager environment



3 Terminology

The following section describes some key terms used throughout this Help.

3.1 Change Data Capture (CDC)

Captures changes in the source data or metadata as they occur and applies them to the target endpoint as soon as possible, in near-real-time. The changes are captured and applied as units of single committed transactions and several different target tables may be updated as the result of a single source commit. This guarantees transactional integrity in the target endpoint. The CDC process for any file or table starts as soon as the data loading operation for the file or table begins.

3.2 Full load

Creates all defined files or tables on the target endpoint, automatically defines the metadata that is required at the target, and populates the tables with data from the source.

3.3 Apply latency

The gap in seconds between capturing a change in one of the source tables and applying that change to the target endpoint.

Latency when applying large transactions

This is best explained by way of example. When the most recent Apply Latency value was 10 seconds and now a transaction of one million rows gets committed at the source endpoint, Replicate starts to apply that transaction to the selected target and it will take some time to write all the changes to the target (for example 60 seconds). During the next 60 seconds, the latency value gradually grows to 70 seconds for the last change in the transaction. Once the transaction is committed, the latency drops back to the 'regular' latency (10 seconds in this case).

Latency when no transactions are being applied

When a time period passes with no changes applied to the target, the latency calculation is based on the time difference between the current time and the timestamp of the last change event read from the transaction log. This could happen, for example, if there is a high volume of activity on tables that were not selected for replication in the current task.

3.4 Source latency

The gap in seconds between when the source database wrote an event to its transaction log and when Replicate captured that change.

3.5 Target latency

The gap between when a commit is seen by Replicate (reading the source transaction log) and when the changes of that commit are seen in the target.

3.6 Overall latency

The overall latency is defined as the time gap between when a change is committed in the source database and when it is visible in the target database.

3.7 Source endpoint

A collection of files or tables managed by an endpoint management system (such as, Oracle, SQL Server) that is part of the main computing service of the IT organization of an enterprise. This source continuously updated, may need to provide a high throughput rate, may have strict 24/7 up-time requirements, and may reference or update a number of tables in the course of a single logical transaction while providing transactional consistency and integrity for the data.

3.8 Target endpoint

A collection of files or tables managed by an Endpoint Management System (DBMS), which may be different from the DBMS managing the source endpoint. It contains data that is derived from the source. It may contain only a subset of the tables, columns, or rows that appear in the source. Its tables may contain columns that do not appear in the source but are transformations or computations based on the source data.

3.9 Net Changes table

Replicate performs data replication based on changes that appear in the source database's transaction log. A single update operation on the source, such as "UPDATE MyTable SET f1=..., f2=..." could potentially update many rows in the source database and create a large number of change records that Replicate will need to apply to the target. Replicate offers two Change Processing modes: **Transactional apply** and **Batch optimized apply**. In **Transactional apply** Change Processing mode, Replicate essentially applies each change to the target, which may take much longer than the original UPDATE took on the source. **Batch optimized apply** mode, on the other hand, is designed to handle efficient replication of a large number of changes. In this mode, Replicate accumulates changes for multiple tables in a memory cache. Repeated changes to the same row are updated in the memory cache. When the maximum memory cache size defined for the task is reached (or when the configured time has elapsed), Replicate does the following:

- a. Writes the cached (net) changes to a special table on the target (the **Net Changes** table),
- b. Bulk uploads the changes to the target table
- c. Uses SQL statements to update the target tables based on the data in the **Net Changes** table.



*Note that for Oracle, Replicate uses a **Net Changes** table per each source table with changes, while for other source endpoints a single **Net Changes** table is used.*

4 Installation and setup

This section describes how to install and set up Qlik Enterprise Manager (Enterprise Manager).



For instruction on installing Enterprise Manager in a Windows Cluster, see [Installing Qlik Enterprise Manager in a Windows cluster \(page 347\)](#).

Enterprise Manager collects information from Qlik Replicate and/or Qlik Compose Servers to allow a central point of monitoring for all Replicate and/or Compose Servers in your organization. Therefore, you also need to install Qlik Replicate and/or Qlik Compose in your organization. For a description of the respective installation procedures, refer to the Qlik Replicate and/or Qlik Compose product Help.

In this section:

- [Preparing your system for Enterprise Manager \(page 15\)](#)
- [Installing Enterprise Manager \(page 18\)](#)
- [Changing the Enterprise Manager service account \(page 24\)](#)
- [Setting the login authentication method \(page 25\)](#)
- [Starting to work with the Qlik Enterprise Manager Console \(page 37\)](#)

4.1 Preparing your system for Enterprise Manager

This section describes the hardware and software requirements for Qlik Enterprise Manager and the monitored Replicate Servers.

In this section:

- [Hardware configuration guidelines \(page 15\)](#)
- [Software requirements \(page 16\)](#)
- [Sizing guidelines \(page 16\)](#)
- [Compatibility with related Qlik Products \(page 17\)](#)
- [Replication Management license \(page 17\)](#)

Hardware configuration guidelines

It is recommended that the machine hosting Qlik Enterprise Manager meets or exceeds the hardware configuration shown in the following table:

Hardware requirements

	Medium System	Large System
Processor	4-core base	16-core base
Memory	8 GB	32 GB
Disk requirements	128 GB	256 GB
Network	1 Gbps	1 Gbps

Sizing guidelines

Depending on the number of Replicate tasks and concurrent users, you might need to balance the system load between multiple Enterprise Manager machines. For example, if your hardware configuration is set up for a large system (40 Replicate servers), it is recommended to monitor no more than 4000 Replicate tasks on a single Enterprise Manager machine. When you near this threshold, then the recommendation is to split the tasks between at least two Enterprise Manager machines by dividing the number of monitored Replicate servers between the Enterprise Manager machines. Similarly, if your hardware configuration is set up for a medium system and the number of Replicate tasks and users concurrently accessing the system is approaching the maximum threshold (see table below), then best practice is to split the monitored Replicate servers between at least two Enterprise Manager machines.

For information on hardware configuration guidelines, see [Hardware configuration guidelines \(page 15\)](#).

Sizing guidelines

	Medium System	Large System
Replicate servers	8	40
Tasks	800	4000
Concurrent users (Rate of public API calls may affect these numbers)	10	80



To some extent, how you set up granular access control might also impact performance. This is usually only a factor when many users are assigned different roles and permissions.

Software requirements



Parts of the software use Java components utilizing OpenJDK JRE, which is included in the Enterprise Manager installation.

Supported Windows platforms

It is strongly recommended to install Qlik Enterprise Manager on a dedicated Windows server, separate from the Replicate and/or Compose Servers that it manages.

Enterprise Manager can be installed on any of the following Windows platforms:

- Windows Server 2016 (64-bit)
- Windows Server 2019 (64-bit)
- Windows Server 2022 (64-bit)

Supported browsers

The following browsers can be used to access the Enterprise Manager Console:

- Microsoft Edge (with automatic updates turned on)
- Mozilla Firefox (with automatic updates turned on)
- Google Chrome (with automatic updates turned on)

Port

Port 443 must be opened for inbound connections on the Enterprise Manager machine.

Additional software requirements

Qlik Enterprise Manager requires the following software:

- Microsoft Visual Studio C++ 2017 x64 Redistributable
- .NET Framework 4.8 or later
- To use the [Analytics](#) feature, PostgreSQL 12.16 or later should be installed either on the Enterprise Manager machine or on a machine that is accessible from Enterprise Manager.
- TLS 1.2 or later must be supported in the underlying OS.



On Windows Server 2012 R2, TLS 1.2 should be turned on by default. If it is not, refer to the Microsoft online help for instructions on how to turn it on.

See also: [Compatibility with related Qlik Products \(page 17\)](#).

Compatibility with related Qlik Products

This version of Enterprise Manager is compatible with the following related products only:

- Qlik Replicate May 2025, Qlik Replicate November 2024, Qlik Replicate May 2024, and Qlik Replicate November 2023 IR and SR1
- Qlik Compose December 2024 (and its Service Releases) only
- Qlik Catalog May 2023 SR2 only

Replication Management license

This section explains how to obtain a Replication Management license and lists the processes that continue even when the license expires or is invalid.

Obtaining a Replication Management license

A Replication Management license is required in order to use Qlik Enterprise Manager. If no license is detected, a **Register License** message will be displayed when you open the Qlik Enterprise Manager Console.



The procedure below does not apply when installing Enterprise Manager in a [High Availability Cluster](#). To obtain a Replication Management license for Enterprise Manager in a High Availability Cluster, you must provide your Qlik Sales Representative with the following information, depending on which Windows version the cluster is running:

- **Windows Server 2016:** The official cluster FQDN.
- **Windows Server 2012 R2:** The FQDN of each of the cluster nodes and the official cluster FQDN.

To obtain a license

1. [Open the Qlik Enterprise Manager Console](#) and copy the Enterprise Manager machine name from either of the following locations:
 - The **Register License** message that is displayed when you open the Qlik Enterprise Manager Console.
 - The bottom of the **Licenses** tab [in the Settings window](#).
2. Request a license from your Qlik Sales Representative, making sure to include the Enterprise Manager machine name in your request.

Process that continue if the license expires or is invalid

The following processes will continue, even if the Replication Management license expires or is invalid:

- Notifications will continue to be sent.
- Tasks monitoring information and messages will continue to be collected from Replicate. However, they will not be visible until a valid Replication Management license is installed.

In such a situation, do one of the following:

- Register a valid Replication Management license.
- Stop the Enterprise Manager service or uninstall the product if you do not intend to use it anymore.

4.2 Installing Enterprise Manager



Enterprise Manager must be installed under an Administrator account.

In this section:

- [Installing Qlik Enterprise Manager using the Setup Wizard \(page 19\)](#)
- [Silently installing Enterprise Manager \(page 21\)](#)
- [Silently upgrading Enterprise Manager \(page 22\)](#)
- [Silently uninstalling Enterprise Manager \(page 23\)](#)

Installing Qlik Enterprise Manager using the Setup Wizard

The following section describes how to install Enterprise Manager.

To install Enterprise Manager:

1. Run the Enterprise Manager setup file (*QlikEnterpriseManager_<version.build>_<systemtype>.exe*, such as *QlikEnterpriseManager_7.0.0.105_X64.exe*). The Enterprise Manager setup wizard opens.
2. Optionally, change the installation directory; then click **Next**.
3. Optionally, change the data directory; then click **Next**. All of the data that is created when you use Enterprise Manager is stored in a directory called **data**. By default, this directory is located in the installation directory where you install Enterprise Manager.



*If you change the data directory location, you must prefix all command line actions with:
-d path_to_the_data_directory*

Example:

```
<product_dir>\bin\AemCt1.exe -d F:\data configuration set -a 123.123.12.1
```

4. Click **Next** again to start the installation.
5. When the installation completes, click **Finish**.
As part of the installation, a new Windows Service called Enterprise Manager is created. The [Enterprise Manager analytics](#) module require a PostgreSQL database. If you plan on using this module, you will need to install PostgreSQL on either the Enterprise Manager machine or on a machine that is accessible from Enterprise Manager. For your convenience, PostgreSQL is included with Enterprise Manager and you will be prompted to install it after clicking **Finish**.
6. Click **Yes** to install PostgreSQL on the Enterprise Manager server or **No** to exit without installing PostgreSQL. You can always install PostgreSQL at a later time by running the PostgreSQL installer from the following location:

```
<Enterprise_Manager_INSTALLATION_FOLDER>\postgreslkit
```

For instructions on installing and maintaining PostgreSQL, refer to the PostgreSQL Help.
7. Open the Enterprise Manager console as described in [Starting to work with the Qlik Enterprise Manager Console \(page 37\)](#).

Upgrading Enterprise Manager

The upgrade process also describes how to back up the Enterprise Manager "data" folder, which will allow you to restore your settings if you encounter any issues with the upgrade.

To upgrade Enterprise Manager:

1. Back up your currently installed version, by copying the Enterprise Manager **data** folder to a location outside the product folder. The default **data** folder location is C:\Program Files\Attunity\Enterprise Manager\data.
2. Run the Enterprise Manager setup wizard to install the new version.

3. If you notice an issue with the upgrade, you can revert to the previous version as described below or do the following:
 - a. Stop all the Enterprise Manager services.
 - b. Overwrite the **data** folder with the **data** folder that you backed up earlier.
 - c. Start all the Enterprise Manager services.

Reverting to a Previous Version

To revert to a previous version:

1. Back up the Enterprise Manager **data** directory to a location outside the product folder.
2. Uninstall the product and then reinstall to the same directory.
3. Once the installation is complete, stop all Enterprise Manager services.
4. Overwrite the **data** directory with the **data** directory that you backed up earlier.
5. Start all Enterprise Manager services.

Migrating the Enterprise Manager Repository

In certain situations, you may need to migrate Replicate or Compose Servers and settings from one Enterprise Manager Server to another. This may be required, for example, if you need to move from a test environment to a production environment or if you need to decommission the Enterprise Manager Server machine. In the migration procedure, which is described below, Server A is the Enterprise Manager Server configured with Replicate or Compose Servers, while Server B is a clean installation of Enterprise Manager Server.



- *All commands should be run as administrator.*
- *The same user must be used for the installation on both servers.*

1. On Server A:
 - a. Run the following commands:
Command 1 - Sets the Master User Key:

```
<INSTALL_DIR>\bin\aemctl.exe [-d data_folder_path] masterukey set -p password
```


where `-d data_folder_path` is only required if the `<INSTALL_DIR>\data` folder is in a non-default location.
Command 2 - Sets the Java Master User Key:

```
<INSTALL_DIR>\java\bin\atajs.bat [-d java_data_folder_path] masterukey set password
```


where `-d java_data_folder_path` is only required if the `<INSTALL_DIR>\data\java` folder is in a non-default location.
 - b. Restart the QlikEnterprise Manager service.
2. On Server B:
 - a. Stop the Qlik Enterprise Manager service.
 - b. Delete the `<INSTALL_DIR>\data` folder.

- c. Copy the **data** folder from Server A to Server B.
- d. Run the following commands:
Command 1:

```
<INSTALL_DIR>\bin\aemctl.exe [-d data_folder_path] masterukey set -p password
```

where `-d data_folder_path` is only required if the `<INSTALL_DIR>\data` folder is in a non-default location.

Command 2:

```
<INSTALL_DIR>\java\bin\atajs.bat [-d java_data_folder_path] masterukey set password
```
- e. Start the QlikEnterprise Manager service.
- f. Log in to Enterprise Manager and verify that the Replicate and Compose Servers have been migrated from Server A to Server B, and that all of Server A's configuration settings have been migrated as well.

Silently installing Enterprise Manager

Enterprise Manager can be installed silently (i.e. without requiring user interaction). This option is useful, for example, if you need to install Enterprise Manager on several machines throughout your organization.



Before commencing the installation, make sure that the prerequisites have been met.

The installation process consists of two stages:

1. [Creating a response file \(page 21\)](#)
2. [Running the silent install \(page 22\)](#)

Creating a response file

Before starting the installation, you need to create a response file.

To create the response file:

1. From the directory containing the Qlik Enterprise Manager setup file, run the following command (note that this will also install Enterprise Manager):

```
QlikEnterpriseManager_version_x64.exe /r /f1my_response_file
```

where:
`my_response_file` is the full path to the response file that will be generated.

Example:

```
QlikEnterpriseManager_6.1.0.536_x64.exe /r /f1C:\Enterprise Manager_install.iss
```



*At the end of the Enterprise Manager installation, when you are prompted to run the PostgreSQL installer, click **No**.*

2. To change the default installation directory, open the response file in a text editor and edit the first **szDir** value as necessary.
3. To change the default data directory, edit the third **szDir** value as necessary.
4. Save the file as **<name>.iss**, e.g. **silent_inst_64.iss**.

Running the silent install

To silently install Qlik Enterprise Manager, open a command prompt and change the working directory to the directory containing the Enterprise Manager setup file. Then issue the following command:

Syntax:

```
QlikEnterpriseManager_version_x64.exe /s /f1my_response_file [/f2log_file]
```

where:

`my_response_file` is the full path to the response file you created earlier and `log_file` is the path to the optional log file.

Example:

```
C:\>QlikEnterpriseManager_6.1.0.536_x64.exe /s /f1C:\temp\1\Enterprise Manager_install.iss /f2C:\temp\1\silent_x64_install.log
```

If the installation was successful, the log file should contain the following rows:

```
[ResponseResult]
ResultCode=0
```

Silently upgrading Enterprise Manager

Silently upgrading Enterprise Manager consists of two stages:

1. [Creating a response file \(page 22\)](#)
2. [Running a silent upgrade \(page 22\)](#)

Creating a response file

Before starting the upgrade, you need to create a response file. You may also want to back up your current installation as described in [Upgrading Enterprise Manager \(page 19\)](#).

For an explanation of how to create a response file, see Step 1 of [Creating a response file \(page 21\)](#).

Running a silent upgrade

To silently upgrade Enterprise Manager, open a command prompt and change the working directory to the directory containing the Enterprise Manager setup file.

Then issue the following command:

Syntax:

```
QlikEnterpriseManager_version_x64.exe /s /f1my_response_file [/f2log_file]
```

where:

`my_response_file` is the full path to the response file you created earlier and `log_file` is the path to the optional log file.

Example:

```
C:\>qlikEnterpriseManager_6.1.0.536_x64.exe /s /f1C:\temp\1\Enterprise Manager_upgrade.iss /f2C:\temp\1\silent_x64_up.log
```

If the upgrade was successful, the log file should contain the following rows:

```
[ResponseResult]
ResultCode=0
```

Silently uninstalling Enterprise Manager

Silently uninstalling Enterprise Manager consists of two stages:

1. [Creating a response file \(page 23\)](#)
2. [Running a silent uninstall \(page 23\)](#)

Creating a response file

Before starting the uninstall, you need to create a response file.

To create the response file:

1. Copy the following (response file) text into a text editor:

```
[{999A7077-16C9-4B3B-AFD2-CBBA9FA72C15}-DlgOrder]
Dlg0={999A7077-16C9-4B3B-AFD2-CBBA9FA72C15}-SdWelcomeMaint-0
Count=3
Dlg1={999A7077-16C9-4B3B-AFD2-CBBA9FA72C15}-MessageBox-0
Dlg2={999A7077-16C9-4B3B-AFD2-CBBA9FA72C15}-SdFinish-0
[{999A7077-16C9-4B3B-AFD2-CBBA9FA72C15}-SdWelcomeMaint-0]
Result=303
[{999A7077-16C9-4B3B-AFD2-CBBA9FA72C15}-MessageBox-0]
Result=6
[{999A7077-16C9-4B3B-AFD2-CBBA9FA72C15}-SdFinish-0]
Result=1
bOpt1=0
bOpt2=0
```
2. Save the file as <name>.iss, e.g. silent_uninst_64.iss.

Running a silent uninstall

To silently uninstall Enterprise Manager, open a command prompt and issue the following command:

Syntax:

```
"C:\Program Files (x86)\InstallShield Installation Information\<directory_containing_setup_file>\setup.exe" /s /f1my_response_file /f2log_file
```

where:

`my_response_file` is the full path to the response file you created earlier and `log_file` is the path to the optional log file.



The directory containing the Enterprise Manager setup file always ends with the following string: `CBBA9FA72C15`

Example:

```
C:\>"C:\Program Files (x86)\InstallShield Installation Information\{999A7077-16C9-4B3B-AFD2-CBBA9FA72C15}\setup.exe" /s /f1C:\temp\response.iss /f2C:\temp\1\silent_uninstall.log
```

If the uninstall was successful, the log file should contain the following rows:

```
[ResponseResult]
ResultCode=0
```

4.3 Changing the Enterprise Manager service account

By default, Enterprise Manager is installed with Administrator privileges. For security reasons, you may want Enterprise Manager to run under a user account that does not have Administrator privileges.

To do this:

1. Install Enterprise Manager.
2. Create a local user without administrative privileges.
3. Reserve the URL for the user you just created by running the following commands:

```
netsh http add urlacl url=https://+:443/AttunityEnterpriseManager
user=DOMAIN\youruser
netsh http add urlacl url=http://+:80/AttunityEnterpriseManager
user=DOMAIN\youruser
```
4. Grant the new user the **Full control** permission for the Enterprise Manager **data** folder (<EM_INSTALL_DIR>\data).
5. Open the **Local Security Policy** window and select **Local Policies>User Rights Assignment**. Then grant the **Log on as a service** policy to the new user.
6. Stop the Enterprise Manager service.
7. In the **Log On** tab of the Enterprise Manager service properties, select **This account** and specify the new user name in the following format:

```
.\NewUser
```
8. Save your changes.
9. Start the Enterprise Manager service.

4.4 Setting the login authentication method

By default, Enterprise Manager uses Single Sign-on through Windows Authentication to authenticate users. This allows users to open the Enterprise Manager Console without providing additional authentication. To require users to authenticate themselves at login, you can change the authentication method to [Form](#) or [SAML](#). Setting the authentication method is done using the Enterprise Manager CLI, as described below.

To see the current authentication settings, run the command described in [Showing the connectivity and login settings \(page 317\)](#).



- All commands in this section should be "Run as administrator" from the Enterprise Manager **bin** directory. The default location is `C:\Program Files\Attunity\Enterprise Manager\bin`.
- When the Enterprise Manager **data** folder is in a non-default location (such as in a cluster installation), make sure to include the `--d data_folder` parameter in all commands, where `data_folder` is the location of the **data** folder. The parameter should immediately follow the name of the Enterprise Manager executable file (e.g. `aemctl --d f:\mydatafolder {command} {parameters}`).
- Changes to the authentication method will take effect only after you restart the Enterprise Manager service.
- If Form authentication is used, all Login/Logout operations are reported to the [Audit Trail](#).

Setting Single Sign-on authentication

This is the default authentication method, which uses Windows authentication.

To set the authentication method to single sign-on, run:

```
aemctl.exe configuration set --authentication_method sso
```

Abbreviated form of --sso: -w

Setting Single Sign-on authentication with Kerberos

Kerberos is an enterprise authentication protocol that uses the concept of tickets and three-way authentication to enable users and computers to identify themselves and secure access to resources.

Using Kerberos SSO, users can seamlessly log into Enterprise Manager and administrators can completely externalize and centrally manage users or group memberships using their existing Kerberos infrastructure.

To set the authentication method to single sign-on with Kerberos, run:

```
aemctl.exe configuration set --authentication_method sso-kerberos
```



If the Kerberos protocol fails, Enterprise Manager will try to log in using NTLM authentication. If NTLM authentication is not enabled in the system, an error will be returned.

Setting form authentication

As opposed to Single Sign-on through Windows Authentication, this method requires users to provide a user name and password at login.

To set the authentication method to Form, run:

```
aemctl.exe configuration set --authentication_method form
```

Abbreviated parameter: `-f`

Setting a user timeout

Parameter: `--user_timeout`

Abbreviated form: `-u`

When setting `--form` authentication, you can use this parameter to override the default user idle timeout period (5 minutes) or to disable user idle timeout entirely.

When a user idle timeout value is specified, Enterprise Manager will automatically log out users that have been inactive for the specified time period (or longer).

To set a user timeout when using Form authentication, run:

```
aemctl.exe configuration set --authentication_method form --user_timeout timeout
```

Where `timeout` is the length of time in minutes after which users will be logged out. The minimum permitted value is 1 minute.

For example, to set a user-idle timeout period of two minutes, run:

```
aemctl.exe configuration set --authentication_method form --user_timeout 2
```

To disable the user-idle timeout entirely, run:

```
aemctl.exe configuration set --authentication_method form --user_timeout -1
```

Specifying an Active Directory domain

Parameter: `--domain`

Abbreviated form: `-m`

When setting `--form` authentication, you can use this parameter to specify an Active Directory domain name that will be used when a user logs in with a user name only (i.e. without a domain name).

To set a user timeout when using Form authentication, run:

```
aemctl.exe configuration set --authentication_method form --domain DomainName
```

Where `DomainName` is the name of the domain.

For example, to set the domain to `ad2_acme`, run:

```
aemctl.exe configuration set --authentication_method form --domain ad2_acme
```

By default, when only a user name is provided in the login form, the domain of the server is used. If the server does not belong to any domain the server machine name will be used instead.

A user who logs in as "doe" will be assumed to identify as "ad2_acme\doe". If a user specifies a fully qualified domain name when logging in, this parameter is ignored.

Setting up SAML authentication

This login method requires you to log in via your organization's SAML Identity Provider. The command parameters for setting SAML authentication are the same regardless of your SAML provider, although the parameter values are slightly different.

The setup procedure consists of the following steps:

- [Step 1: Set up SAML on Enterprise Manager \(page 28\)](#)
- [Step 2: Set Up an Enterprise Manager Superuser \(page 29\)](#)
- [Step 3: Log in to Enterprise Manager and create SAML users \(page 29\)](#)



Before running the commands, you must have already configured Enterprise Manager as an application in your SAML Identity Provider.

When Enterprise Manager is not installed in a Cluster, the Enterprise Manager Assertion Consumer Service (ACS) URL is:

`https://{host_name}/attunityenterprisemanager/rest/?action=login_saml`

When Enterprise Manager is installed in a Cluster, make sure to fulfill the following prerequisites:

- *Finish the cluster install on all nodes before configuring SAML.*
- *In order to propagate the configuration changes, make sure to include the `-d data_folder` parameter in the SAML configuration commands described below, where `data_folder` is the location of the cluster's shared **data** folder.*
- *The Assertion Consumer Service (ACS) URL which the IDP should call when redirecting SAML calls should be the cluster wide DNS name (as opposed to a specific machine name).*

This is how it should look:

`https://{em-cluster-fqdn}/attunityenterprisemanager/rest/?action=login_saml`

For more information about setting up Enterprise Manager in a cluster, see [Installing Qlik Enterprise Manager in a Windows cluster \(page 347\)](#).

Step 1: Set up SAML on Enterprise Manager

To set the Enterprise Manager authentication method to SAML, run:

Syntax:

```
aemctl [--d data_folder] configuration set --authentication_method SAML --idp_url ssourl --idp_issuer issuer_name --idp_certificate_file CertificateFile [--idp_user_displayname_attribute displayname] [--idp_username_attribute username] [--idp_user_groups_attribute groups]
```

Example: Using Microsoft Azure as the SAML IdP

```
aemctl configuration set --authentication_method SAML --idp_url https://login.microsoftonline.com/12854727-3c42-4866-ab29-0c418b8310a1/saml2 --idp_issuer aemdevtest --idp_certificate_file AEMDevTest.pem
```

Where:

- *ssourl* is the URL of the SAML IdP (Identity Provider) that handles sign-in requests. When using Okta, this is the Okta Single Sign-On URL. When using Microsoft Azure, this is the Azure AD SAML single sign-on URL. Enterprise Manager will direct users to this URL to complete the SAML login operation.
- *issuer_name* is a unique name that the identity provider uses for SAML 2.0. When using Okta, this should be a URL. When using Azure, this should be a string.
- *certificateFile* - The certificate is used by the IdP to sign the SAML assertions sent to Enterprise Manager. The certificate file can be anywhere on the Enterprise Manager machine, but only needs to be specified with a path when it does not reside in the Enterprise Manager **bin** directory. When using Okta, the certificate must be in **.cert** format. When using Microsoft Azure, the certificate must be in **.pem** format.
- *data_folder* - The location of the data folder when it is not the default location such as when Enterprise Manager is [installed in a Windows cluster](#).

Optional Parameters:

The following parameters are optional and should only be used if required by your SAML IdP:

- *--idp_user_displayname_attribute* - The user display name attribute.
- *--idp_username_attribute* - By default, with SSO, the SAML Assertion's "Subject" attribute is used to define the username. Using the subject is the right solution in most situations, but in extreme cases (such as the subject being a transient ID) it may be necessary to provide the username in some other form.
- *--idp_user_groups_attribute* - The user group attribute.



Once you have set up SAML, you need to restart the Qlik Enterprise Manager service for the settings to take effect.

Step 2: Set Up an Enterprise Manager Superuser

The first time you log in to Enterprise Manager using SAML, you must log in as an Enterprise Manager superuser. This is because none of the existing (or default) Enterprise Manager users are authorized SAML users.



The instructions below assume that you have already changed the default Master User key. For instruction on how to do this, see [Generating a random Master User Password \(page 315\)](#) and [Setting or changing the MUK \(Master User Key\) \(page 315\)](#) respectively.

To set up a superuser, run the following command:

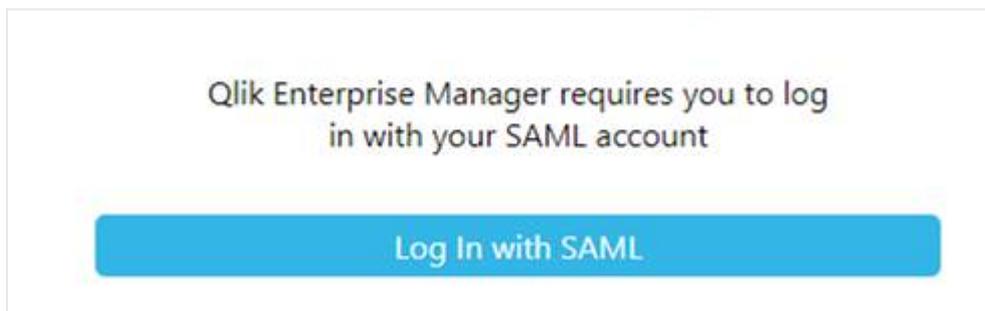
```
aemctl authorization setsuperuser -s username -e ExpirationTimeoutInMinutes -m MasterUserKey
```

Where:

- *username* is the superuser user name. The user must be an existing SAML user and can contain any Unicode character up to 255 characters.
- *ExpirationTimeInMinutes* is the expiration time for the specified user. The maximum is 60 minutes.
- *MasterUserKey* is your Master User Key.

Step 3: Log in to Enterprise Manager and create SAML users

After setting up SAML authentication, you will be presented with the following page when you try to open the Qlik Enterprise Manager console:



1. Click **Log In with SAML**.
You will be redirected to Okta or Microsoft Azure to provide your SAML login credentials.
2. Authenticate yourself with SAML.
After successful authentication, you will be redirected back to the Enterprise Manager Console.
3. Add authorized SAML users and groups, as described in [Managing user permissions \(page 307\)](#).

For information about other CLI options, see [Configuring Enterprise Manager using the CLI \(page 313\)](#).

Switching between SAML and SSO/Form authentication

If you wish to switch from SAML to SSO/Form authentication (or vice versa), you may need to clear the existing users from Enterprise Manager. When switching from SAML to SSO/Form authentication, you will not be able to change user permissions or add users unless you clear the existing SAML users. However, when switching from SSO/Form authentication to SAML, you do not need to clear the existing users in order to add/delete users or edit user permissions. You may still want to do this however if you find the presence of non-SAML users distracts you from managing the SAML users.



Before clearing the users, it is strongly recommended to export them as this will save time (by importing them) should you later need to revert the authentication type.

To export all users to a JSON file, run the following command:

```
aemctl repository export_acl -f [fullpath\]filename
```

where `filename` is the name of the file to which the users will be exported. By default, the file will be exported to the Enterprise Manager **bin** directory. If you wish the file to be exported to a custom path, include `[fullpath\]` in the command.

Example:

```
aemctl repository import_acl -f C:\temp\Enterprise ManagerUsers
```

To clear all users, run the following command:

```
aemctl repository clear_acl
```

To import users from the exported JSON file, run the following command:

```
aemctl repository import_acl -f [fullpath\]filename
```

where `filename` is the name of the file to import. Unless you specified a custom export path, the file will be located in the Enterprise Manager **bin** directory. If you wish the file to be imported from a custom path, include `[fullpath\]` in the command.

Example:

```
aemctl repository import_acl -f C:\temp\Enterprise ManagerUsers
```

Setting up Personal Access Token authentication for the API

Personal Access Token authentication is only supported with the Enterprise Manager API.

The high-level flow consists of the following steps:

- Step 1: In Okta or Azure Active Directory, create an app integration that uses OpenID Connect
- Step 2: In Enterprise Manager, enable Enterprise Manager to communicate with your IdP (Okta or Azure Active Directory).

- Step 3: In Enterprise Manager, generate a Personal Access Token
- Step 4: Configure the Enterprise Manager API to log in using the Personal Access Token

Step 1: Create an app integration that uses OpenID Connect

Enterprise Manager uses OpenID Connect to log in with the API. Therefore, before you can use OpenID Connect with Enterprise Manager, you must create a web integration in your IdP.

Creating a web integration in Okta

To create a web integration in Okta for use with Enterprise Manager:

1. Log in to your Okta account.
2. Navigate to **Applications>Applications** and click **Create App Integration**.
3. In the **Create a new app integration** dialog, select **OIDC - Open IDConnect**.
4. Select **Web Application** as the **Application type** and click **Next**.
5. In the **New Web Integration** page, configure the following fields:
 - **App Integration Name:** The name of your App integration. For example, **QEM OpenID Connect**.
 - Select **Refresh Token**.
 - In the **Sign-in redirect URIs** field, enter the following:

```
https://EnterpriseManagerMachine/attunityenterprisemanager/rest/login_openid
```

Where `EnterpriseManagerMachine` is the host name or IP address of your Enterprise Manager machine.
6. Clear the **Enable immediate access with Federation Broker Mode** option, and then click **Save**.
7. Copy your **Client ID** and **Client secret**. You will need to provide these parameters in the next stage.
8. Assign the app integration to the users or groups that you want to allow to use the Personal Access Token.
For details, see [≤ Assign app integrations](#).

Creating a web integration in Azure Active Directory

To create a web integration in Azure Active Directory for use with Enterprise Manager:

1. Log in to Azure Portal (<https://portal.azure.com/>)
2. Navigate to Microsoft Entra ID.
3. In the **Manage** menu on the left, select **Enterprise applications**.
4. In the **Enterprise applications | All applications** screen, click **New Application**.
5. Under **Browse Microsoft Entra Gallery**, click **Create your own application**.
6. Name your application and choose one of the available options under **What are you looking to do with your application?**
7. Click **Create**.
8. In the **Manage** menu on the left, select **App registrations**.
 - a. Make sure the **All applications tab** is selected on the right.
 - b. Select the app you created earlier.

- c. Click **Redirect URIs**.
 - d. Click **Add a platform**.
 - e. Select **Web**.
 - f. Enter the **Redirect URI**:
`https://EnterpriseManager/attunityenterprisemanager/rest/login_openid`
Where *EnterpriseManager* is replaced with the host name or IP address of your Enterprise Manager machine
 - g. Copy your **Client ID** (under Overview) and your **Client secret** (under Certificates & secrets). You will need to provide these parameters in the next stage.
9. Assign the app integration to the users or groups that you want to allow to use the Personal Access Token.
For instructions how to do this, see [Assign a user account to an enterprise application](#).

Step 2: Enable Enterprise Manager to communicate with your IdP

Enabling Enterprise Manager to communicate with Okta

To enable Enterprise Manager to communicate with Okta, open a CMD prompt as admin and change the working directory to `<Enterprise Manager-INSTALL-DIR>\bin`.

Then run the following command:

Syntax

```
aemctl.exe configuration set --open_id_authority your-openid-connect-authority --open_id_client_id your-client-id --open_id_client_secret your-secret [OPTIONAL PARAMETERS]
```

Required parameters

- `--open_id_authority` is your Okta URL. For example, `https://dev-13465054.okta.com`
- `--open_id_client_id` is the client ID generated in **Step 1: Create an app integration that uses OpenID Connect** above.
- `--open_id_client_secret` is the client secret generated in **Step 1: Create an app integration that uses OpenID Connect** above.

Optional parameters

The following parameters are optional and should only be used if required by Okta:

- `--open_id_additional_scopes` - Additional scopes that are added to the scope list when an OpenID Connect login occurs. The default is "groups"
- `--api_token_daily_maintenance_time` - Determines when the API token maintenance background process runs each day. This should be formatted as HH:mm. The default is "00:30"
- `--api_token_lifetime` - The number of days a Personal Access Token is valid. The default is "180"
- `--open_id_refresh_token_lifetime` - The number of days a refresh token is valid. The default is "0" meaning it is valid forever.

- `--open_id_user_name_field` - The field name for the OpenID Connect user name. The default is "preferred_username".
- `--open_id_display_name_field` - The field name for the OpenID Connect user display name. The default is "name".
- `--open_id_group_field` - The field name for an OpenID Connect group. The default is "groups".
- Parameters for enabling WinHTTP proxy connectivity:
 - `enable_winhttp_proxy` - Enable WinHTTP proxy
 - `proxy_user_name` - The WinHTTP proxy username. To disable proxy authentication, specify `--proxy_user_name ""` (empty quotation marks)
 - `proxy_password` - The WinHTTP proxy password.

Example

```
aemctl.exe configuration set --open_id_authority "https://dev-13465054.okta.com" --open_id_client_id "00a8ohk15ftweZNWTT5d7" --open_id_client_secret "FJxxQwopJsrOGrthsavzfUICnthG6JLA1-CTHUJO"
```



After you have run the OpenID Connect command, you need to restart the Qlik Enterprise Manager service for the settings to take effect.

Enabling Enterprise Manager to communicate with Azure Active Directory

To enable Enterprise Manager to communicate with Azure Active Directory, open a CMD prompt as admin and change the working directory to `<Enterprise Manager-INSTALL-DIR>\bin`.

Then, run the following command:

Syntax

```
aemctl.exe configuration set --open_id_authority your-openid-connect-authority --open_id_client_id your-client-id --open_id_client_secret your-secret [OPTIONAL PARAMETERS]
```

Required parameters

- `--open_id_authority` is your Azure AD URL. For example, `https://login.microsoftonline.com/146d3649-0e6e-4584-af13-1063888e4915/v2.0`
- `--open_id_client_id` is the client ID generated in **Step 1: Create an app integration that uses OpenID Connect** above.
- `--open_id_client_secret` is the client secret generated in **Step 1: Create an app integration that uses OpenID Connect** above.

Optional parameters

The following parameters are optional and should only be used if required by Azure Active Directory:

- `--open_id_issuer` - The issuer specified in the OpenID Connect Discovery document. This parameter is only required if the issuer is different from the `open_id_authority` specified in the command.
- `--open_id_trusted_url_prefixes` - A list of space-separated trusted URL prefixes that appear in the OpenID Connect Discovery document.
- `--open_id_claims_source` - Instructs Enterprise Manager how to retrieve the claims for a user connecting with OpenID Connect via the public API. The value can be `UserInfo` or `IdentityToken` (case insensitive). The default is `UserInfo`, but you must specify `IdentityToken` for Azure.
- `--open_id_additional_scopes` - Additional scopes that are added to the scope list when an OpenID Connect login occurs. The default is "groups".
- Parameters for enabling WinHTTP proxy connectivity:
 - `enable_winhttp_proxy` - Enable WinHTTP proxy
 - `proxy_user_name` - The WinHTTP proxy username. To disable proxy authentication, specify `--proxy_user_name ""` (empty quotation marks)
 - `proxy_password` - The WinHTTP proxy password.

Example

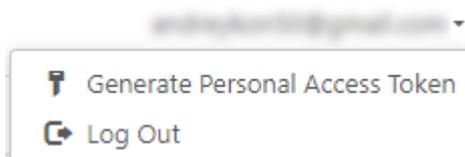
```
aemctl configuration set --open_id_authority https://login.microsoftonline.com/146d3649-0e6e-4584-af13-1063888e4915/v2.0 --open_id_client_id a8883e53-528f-4fa3-956a-5997dca94cba --open_id_client_secret XUDF5~WERGFWRE8J554230959TJI9540 --open_id_issuer https://login.microsoftonline.com/146d3649-0e6e-4584-af13-1063888e4915/v2.0 --open_id_trusted_url_prefixes https://login.microsoftonline.com https://graph.microsoft.com/oidc/userinfo --open_id_additional_scopes "" --open_id_claims_source IdentityToken
```



After you have run the OpenID Connect command, you need to restart the Qlik Enterprise Manager service for the settings to take effect.

Step 3: In Enterprise Manager, generate a Personal Access Token

1. Log in to Enterprise Manager as a SAML user. This must be one of the users/groups that was assigned to the app integration in **Step 1: Create an app integration that uses OpenID Connect** above.
For information on setting up SAML, see **Setting up SAML authentication** above.
2. In the top right of Enterprise Manager, click the inverted triangle to the right of the user name and select **Generate Personal Access Token**.



The **Generate Personal Access Token** dialog opens.

In the **Generate Personal Access Token** dialog, you will see one of the following:

- **You do not have a Personal Access Token.** This is shown if you have not previously generated a Personal Access Token:

- **Your Personal Access Token expired on <Date>**. This is shown if your Personal Access Token has expired.
- **Your Personal Access Token expires on <Date>**. This is shown if you already have a Personal Access Token.



When regenerating a token, you will need to confirm that you want to regenerate the token. This is because API login with the original token will stop working as soon as you generate a new token.

3. Click **Generate token**.
The **Copy Personal Access Token** dialog is displayed.
4. Copy your personal access token. You will need this to log in with the Enterprise Manager API.

See also: [Managing Personal Access Tokens \(page 309\)](#)

Step 4: Configure the Enterprise Manager API to log in using the Personal Access Token

For instructions, see:

- REST API: [Login](#)
- .NET API: [Getting started - Login](#)
- Python API: [Getting started - Login](#)

Logging in to Enterprise Manager with a JSON Web Token via the API

You can log in to Enterprise Manager using a JWT (JSON Web Token) from your IDP. When using this login method:

- The user program retrieves the JWT from the IDP.
- Enterprise Manager validates it and returns a session ID for the program to use via the Enterprise Manager API.

This method involves setting up Enterprise Manager to work with the JWT and then logging in using the Enterprise Manager API.

Step 1: Set up Enterprise Manager to work with the JWT

This step involves setting up Enterprise Manager to do the following:

- a. Get the JWKS (JSON web keys) from the IDP
- b. Validate the JWT
- c. Extract the user identity and groups from the JWT

To do this:

Open a CMD prompt as admin and change the working directory to `<Enterprise Manager-INSTALL-
DIR>\bin`.

Then run the following command:

Syntax

```
aemctl.exe configuration set --jwt_user_name_field non-default-value --jwt_group_field non-default-value --jwt_authority value --jwt_audience value --jwt_clock_skew_seconds non-default-value [OPTIONAL PARAMETERS]
```

Example command

```
aemctl.exe configuration set --jwt_user_name_field myname --jwt_authority https://login.microsoftonline.com/146d3649-0e6e-4584-af13-1063888e4915/v2.0 --jwt_audience 94cf7233-6a93-4858-aa0f-a4f2a491b01c --jwt_clock_skew_seconds 0 --jwt_issuer https://login.microsoftonline.com/146d3649-0e6e-4584-af13-1063888e4915/v2.0 jwt_trusted_url_prefixes https://login.microsoftonline.com https://sts.windows.net https://graph.microsoft.com
```

Required parameters

These parameters are used to retrieve the JWKS (JSON Web Key Set) from the IDP and validate the JWT:

- `jwt_clock_skew_seconds` - The allowed time difference (in seconds) between the server and client clocks when validating the "nbf" and "exp" fields in the JSON Web Token. The default is 300 seconds.
- `jwt_authority` - The authority used for JSON Web Token validation.
- `jwt_audience` - The audience in the JSON Web Token's "aud" field.

These parameters are used to extract the user information from the JWT:

- `jwt_user_name_field` - The name of the JSON Web Token claim containing the username. The default is "preferred_username".

Optional parameters

The following parameters are optional and should only be used if required by the IdP or by your corporate security policies:

- `jwt_group_field` - The name of the JSON Web Token claim containing the group names. The default is "groups".
- `jwt_issuer` - The issuer in the JSON Web Token's "iss" field, and in the OpenID Connect metadata document's "issuer" field. This parameter is only required if the issuer is different from the `jwt_authority`.
- `jwt_trusted_url_prefixes` - Space-separated trusted URL prefixes in the OpenID Connect Discovery document.
- Parameters for enabling WinHTTP proxy connectivity:
 - `enable_winhttp_proxy` - Enable WinHTTP proxy
 - `proxy_user_name` - The WinHTTP proxy username. To disable authentication, specify `--proxy_user_name ""` (empty quotation marks)
 - `proxy_password` - The WinHTTP proxy password.

Step 2: Configure the Enterprise Manager API to log in using the JWT

For instructions, see:

- REST API: [Login](#)
- .NET API: [Getting started - Login](#)
- Python API: [Getting started - Login](#)

4.5 Starting to work with the Qlik Enterprise Manager Console

To start working with Enterprise Manager, you need to open the Qlik Enterprise Manager Console and [register a Replication Management license](#).

You can use a Web browser to access the Console from any computer in your network. For information on supported browsers, see [Preparing your system for Enterprise Manager \(page 15\)](#).



The user logged into Enterprise Manager must be an authorized Qlik Enterprise Manager user.

To access the Qlik Enterprise Manager Console:

- From the machine on which it is installed, select **All Programs > Qlik Enterprise Manager > Qlik Enterprise Manager Console** from the **Windows Start** menu.
OR
Type the following address in the address bar of your Web browser:
https://<computer name>/attunityenterprisemanager
On a machine running Microsoft Windows 10 or Windows Server 2012, you need to run the Console as Administrator.
- From a remote browser, type the following address in the address bar of your Web browser:
https://<computer name>/attunityenterprisemanager
where <computer name> is the name or IP address of the computer where Qlik Enterprise Manager is installed.

If no server certificate is installed on the Enterprise Manager machine, a page stating that the connection is untrusted opens. This is because when Enterprise Manager detects that no server certificate is installed, it installs a self-signed certificate. Because the browser has no way of knowing whether the certificate is safe, it displays this page.

For more information, see [Setting up HTTPS for the Enterprise Manager console \(page 39\)](#).

If prompted, enter your user name and password.

The user name may need to include domain information in the following format:

<domain name>@<user name>

For more information, see [Setting the login authentication method \(page 25\)](#).

Registering Licenses

If this is the first time you are using Enterprise Manager, you will be prompted to register a Replication Management license when the console opens. You may also need to register a Replication Analytics license (required for the [Analytics \(page 325\)](#) module), depending on whether you have obtained such a license from your Qlik Sales Representative.

For information on registering licenses, see [Registering and managing licenses \(page 296\)](#).

5 Security considerations

This section provides a detailed rundown of the various security-related procedures that need to be performed to ensure that your data is secure.

In this section:

- [Setting up HTTPS for the Enterprise Manager console \(page 39\)](#)
- [Setting up HSTS on Enterprise Manager \(page 42\)](#)
- [Replacing the self-signed certificate on Windows \(page 43\)](#)
- [Setting the hostname and changing the SSL port \(page 44\)](#)
- [Replacing the Master User Password \(page 45\)](#)
- [Encrypting the user permissions file \(page 46\)](#)
- [Controlling execution of user-defined commands \(page 47\)](#)

5.1 Setting up HTTPS for the Enterprise Manager console

Industry-standard security practices dictate that web user interface for enterprise products must use secure HTTP (HTTPS). Qlik Enterprise Manager enforces the use of HTTPS and will not work if HTTPS is configured incorrectly.

As Enterprise Manager uses the built-in HTTPS support in Windows, it relies on the proper setup of the Windows machine it runs on to offer HTTPS access. In most organizations, the IT security group is responsible for generating and installing the SSL server certificates required to offer HTTPS. It is strongly recommended that the machine on which Enterprise Manager is installed already has a valid SSL server certificate installed and bound to the default HTTPS port (443).

Checking if an SSL Certificate is installed

To check whether an SSL certificate is installed, you can use the following command:

```
netsh http show sslcert | findstr /c:":443"
```

If an SSL certificate is installed, the output should look like this:

```
netsh http show sslcert | findstr /c:":443 "  
IP:port : 192.168.1.13:443  
IP:port : 192.168.1.11:443  
IP:port : [fe80::285d:599c:4a55:1092%11]:443  
IP:port : [fe80::3d0e:fb1c:f6c3:bc52%23]:443
```

With a valid SSL certificate installed, the Enterprise Manager web user interface will automatically be available for secure access from a web browser using the following URL:

<https://<machine-name>/attunityenterprisemanager>

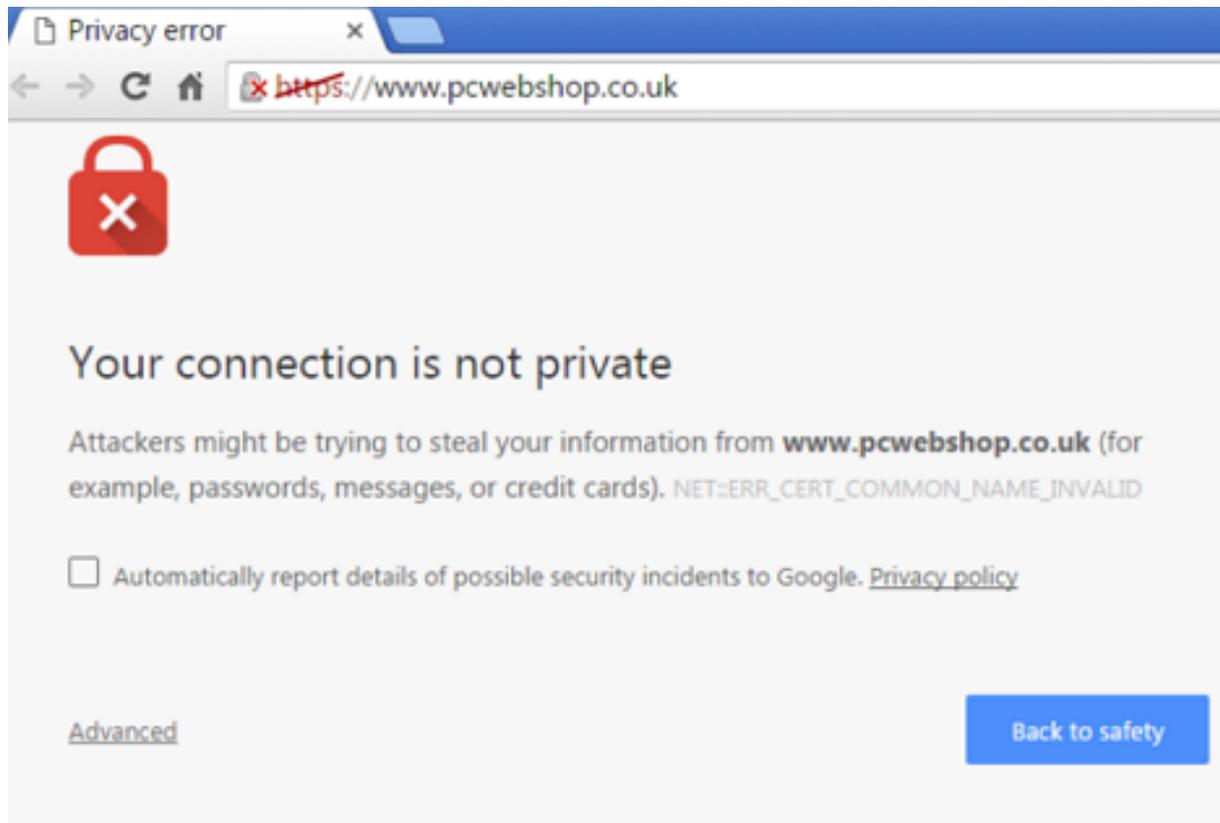
Using the self-signed certificate

Due to the way the HTTPS protocol works, there is no way for Enterprise Manager to automatically provide and install a valid SSL server certificate. Still, in the event that no SSL server certificate is installed, Enterprise Manager automatically generates and installs a self-signed SSL server certificate (as a

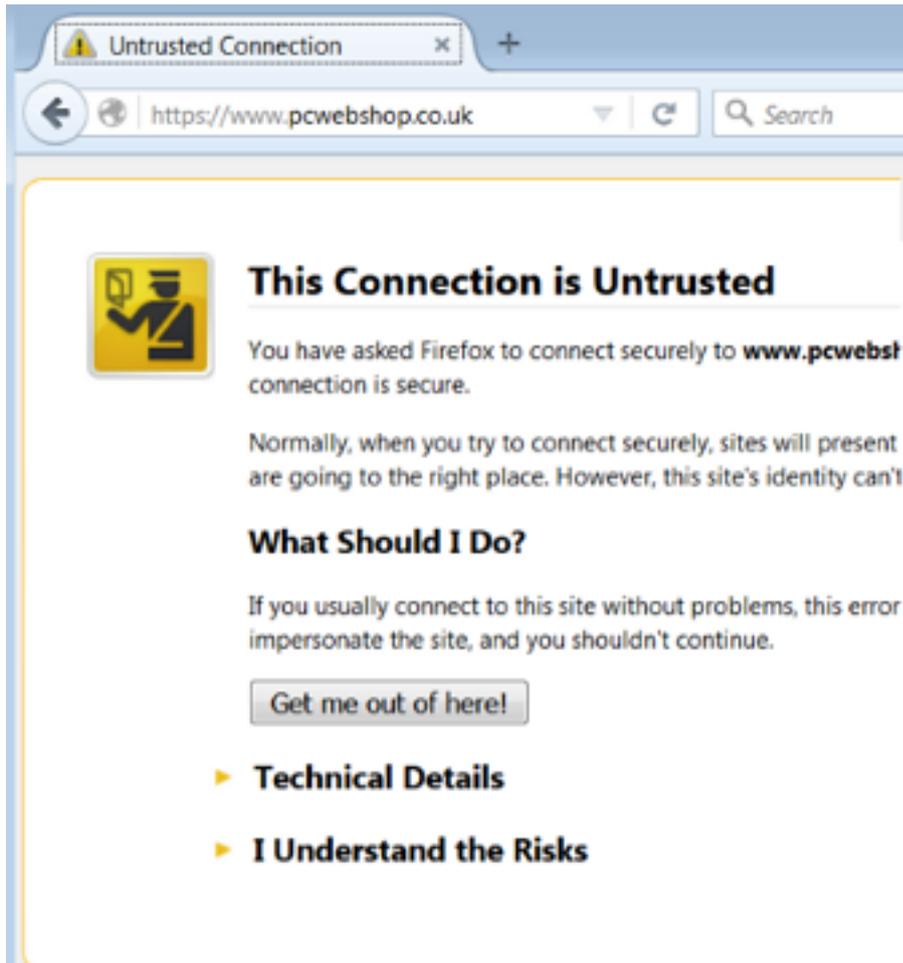
5 Security considerations

temporary measure). This certificate is generated on the Enterprise Manager machine and cannot be exported or used elsewhere.

It should be noted that browsers do not consider the certificate to be valid because it was not signed by a trusted certificate authority (CA). When connecting with a browser to a server that uses a self-signed certificate, a warning page is shown such as this one in Chrome:



Or this one in Firefox:



The warning page informs you that the certificate was signed by an unknown certificate authority. All browsers display a similar page when presented with a self-signed certificate. If you know that the self-signed certificate is from a trusted organization, then you can instruct the browser to trust the certificate and allow the connection. Instructions on how to trust the certificate vary between browsers and even between different versions of the same browser. If necessary, refer to the help for your specific browser.



Some corporate security policies prohibit the use of self-signed certificates. In such cases, it is incumbent upon the IT Security department to provide and install the appropriate SSL server certificate (as is the practice with other Windows products such as IIS and SharePoint). If a self-signed certificate was installed and needs to be removed, then the following command can be used:

```
<product_dir>\bin\AemCtl.exe certificate clean
```

Note that after the self-signed certificate is deleted, connections to the Enterprise Manager machine will not be possible until a valid server certificate is installed. Should you want to generate a new self-signed certificate (to replace the deleted certificate), simply restart the Enterprise Manager service.

5.2 Setting up HSTS on Enterprise Manager

HSTS is a web security policy mechanism that helps to protect websites against man-in-the-middle attacks such as protocol downgrade attacks and cookie hijacking. It allows web servers to declare that web browsers (or other complying Dilqam) should automatically interact with it using only HTTPS connections, which provide Transport Layer Security (TLS/SSL).

You can force the Enterprise Manager Web UI and/or the Enterprise Manager REST API connections to use HSTS (HTTP Strict Transport Security). To do this, run the commands described below.

All commands should be run from as Admin from the product **bin** folder.

Enabling HSTS

Command syntax

```
aemctl.exe configuration set --static_http_headers header_list --rest_http_headers header_list
```

Parameters

Parameter	Description
--static_http_headers	The headers required to connect to the Enterprise Manager Web UI.
--rest_http_headers	The headers required to connect using the API.

Headers should be specified using the following format:

```
aemctl.exe configuration set --static_http_headers "header1:value1" "header2:value2" --rest_http_headers "header1:value1" "header2:value2"
```

Example

```
aemctl.exe configuration set --static_http_headers "Strict-Transport-Security:max-age=31536000; includeSubDomains;" --rest_http_headers "Strict-Transport-Security:max-age=31536000; includeSubDomains;"
```

Disabling HSTS

You can also revert to regular HTTPS connections.

Command syntax

```
aemctl.exe configuration set --static_http_headers ""|--rest_http_headers ""
```

Parameters

Parameter	Description
--static_http_headers	Use this parameter to revert the headers required to connect to the Enterprise Manager Web UI.
--rest_http_headers	Use this parameter to revert the headers required to connect using the API.

Example

Disable static_http_headers

```
aemctl.exe configuration set --static_http_headers ""
```

Disable rest_http_headers

```
aemctl.exe configuration set --rest_http_headers ""
```

5.3 Replacing the self-signed certificate on Windows

The instructions below are intended for organizations who wish to replace the built-in self-signed certificate automatically generated by the Enterprise Manager UI Server on Windows with their own certificate. This is achieved by removing the self-signed certificate and then importing the new certificate.

See also [Setting up HTTPS for the Enterprise Manager console \(page 39\)](#).

Before starting, make sure that the following prerequisites have been met:

- The replacement certificate must be a correctly configured SSL PFX file containing both the private key and the certificate.
- The common name field in the certificate must match the URL name that browsers will use to access the Enterprise Manager Console.
- The Certificate Subject Alternative Name must match the URL name that browsers will use to access the Enterprise Manager Console.

To remove the self-signed certificate created by Enterprise Manager:

1. Stop the Qlik Enterprise Manager service.
2. Open a command prompt (using the "Run as administrator" option) and change the path to the **bin** directory. The default path is:
C:\Program Files\Qlik\Enterprise Manager\bin.
3. Run the following command:
`AemCtl.exe certificate clean`

To import your own certificate:

1. Run `mmc.exe` to open the Microsoft Management Console.
2. From the **File** menu, select **Add/Remove Snap-in**.
The **Add or Remove Snap-ins** dialog box opens.
3. In the left pane, double-click **Certificates**.
The **Certificates snap-in** wizard opens.
4. Select **Computer account** and then click **Next**.
5. In the **Select Computer** screen, make sure that **Local computer** is selected and then click **Finish**.
6. Click **OK** to close the **Add or Remove Snap-ins** dialog box.
7. In the left pane, expand the **Certificates** folder. Then, right-click the **Personal** folder and select **All Tasks>Import**.

8. In the **File to Import** screen, select your PFX certificate file. Note that by default the **Open** dialog box displays CER files. In order to see your PFX files, you need to select **Personal Information Exchange** from the drop-down list in the bottom right of the dialog box.
9. Click **Next** and enter the private key password.
10. Continue clicking **Next** until you reach the **Completing the Certificate Import Wizard** screen. Then click **Finish** to exit the wizard.
11. In the **Personal > Certificates** folder, double-click the newly imported certificate. The **Certificate** dialog box opens.
12. Scroll down the **Details** tab until you see the **Thumbprint** details and copy them to the clipboard.
13. Open a command prompt and run one of the following commands:

Syntax:

```
⌘ netsh http add sslcert ipport=0.0.0.0:443 certhash=[YOUR_CERTIFICATE_THUMBPRINT_WITHOUT_SPACES] appid={4dc3e181-e14b-4a21-b022-59fc669b0914}
```

Example:

```
netsh http add sslcert ipport=0.0.0.0:443  
certhash=5f6eccba751a75120cd0117389248ef3ca716e61 appid={4dc3e181-e14b-4a21-b022-  
59fc669b0914}
```

Syntax:

```
⌘ netsh http add sslcert ipport=[::]:443 certhash=[YOUR_CERTIFICATE_THUMBPRINT_WITHOUT_SPACES] appid={4dc3e181-e14b-4a21-b022-59fc669b0914}
```

Example:

```
netsh http add sslcert ipport=[::]:443 certhash=5f6eccba751a75120cd0117389248ef3ca716e61  
appid={4dc3e181-e14b-4a21-b022-59fc669b0914}
```

14. Close the command prompt and Microsoft Management Console.
15. Start the Qlik Enterprise Manager service.

5.4 Setting the hostname and changing the SSL port

After installing Qlik Enterprise Manager, you can use the Enterprise Manager CLI to set the hostname and SSL port for accessing the Enterprise Manager server machine.

Under normal circumstances, you should not need to set the hostname. However, on some systems, connecting using HTTPS redirects to localhost. If this occurs, set the hostname of the Enterprise Manager machine by running the command shown below.

To set the hostname:

Run the following command:

```
<product_dir>\bin\AemCtl.exe configuration set -a hostAddress  
where hostAddress is the address of the Enterprise Manager server machine.
```



When using a Load Balancer, hostAddress should be the Load Balancer host address.

To change the SSL port:

Run the following command:

```
<product_dir>\bin\AemCt1.exe configuration set -s httpsPort
```

where httpsPort is the SSL port number of the Enterprise Manager server machine.

5.5 Replacing the Master User Password

Qlik goes to great lengths to ensure that sensitive information is protected from unauthorized parties, which is why all passwords and other sensitive information are encrypted (and decrypted) using the Enterprise Manager Master Key. Enterprise Manager uses only FIPS 140-2 compliant algorithms for hashing and encryption: SHA256 for hashing and AES256 for encryption.

This topic provides an overview of how the Enterprise Manager Master Key is generated and applied as well as explaining how to encrypt the User Permissions file.

The Master Key

The following section describes the role the master key and the master user key play in keeping your data secure.

- Using the Enterprise Manager CLI, the user provides the system with an alphanumeric password (Master User Password), which must be at least 32 characters.



By default (i.e. after first time installation), the Master User Password is randomly generated by Enterprise Manager. It is strongly recommended to change the Master User Password as soon as possible (especially in a production environment) as this will allow recovery of backed up data in the event of disk failure. The password should of course be kept in a secure location for future use. You can either use your own password or run the `genpassword` utility to generate a password for you.

- Enterprise Manager uses a one way hashing functions to generate a key (Master User Key) from the Master User Password.



The one way hash function guarantees that, given the same Master User Password as input, the same Master User Key will be generated. This is what happens in [high availability scenarios](#) - the same Master User Password is entered on two different machines.

- Enterprise Manager encrypts the Master User Key using the local machine's key (Windows) and stores it in a file named `muk.dat`. The `muk.dat` file can contain several entries (and indeed does in a

[high availability environment](#)), with each entry uniquely identifying the machine on which Enterprise Manager is running.

- Enterprise Manager generates a random Master Key, encrypts it with the Master User Key and stores it in a common location (e.g. the root repository).
- On software startup, the following occurs:
 - The Master User Key is read and decrypted (using the local machine's key)
 - The Master Key is read and decrypted using the Master User Key

Once this process is complete, Enterprise Manager is able to encrypt and decrypt all sensitive information.

For more information on commands related to the master user password, see [Commands Related to the Master User Password](#).

High Availability mode

In a High Availability environment, Enterprise Manager runs on at least two machines that share the same data folder. As there are two machines, there will also be two different keys - one for each machine.

As mentioned earlier, the Master User Key file (*muk.dat*) is constructed of entries, with each entry corresponding to one machine. Both machines scan the entries searching for an entry they can decrypt.

If no matching entry can be found, an error will be returned. In this case, simply use the Enterprise Manager CLI to enter the Master User Password again, and create an entry for the new machine.

See also [Installing Qlik Enterprise Manager in a Windows cluster \(page 347\)](#).

5.6 Encrypting the user permissions file

User permissions are stored in the following repository file:

```
<product_dir>\Data\cfgrepo.sqlite
```

To prevent unauthorized access of this file, you can encrypt it using the procedure described below. After you perform the procedure, the repository file will be encrypted with the AES-256 bit cipher.



The length of any passwords specified during the procedure must be at least 32 characters.

To encrypt the repository file:

1. Open a command prompt as administrator and change the working directory to:
`<product_dir>\bin`
2. Run the following command to set the master user key:
`aemctl.exe masterukey set --password your_MasterUserPassword`
Example:
`aemctl.exe masterukey set --password ANqaGYERP3UKmGLK6UNuMqrkAGxwH8FM`
3. Restart the Qlik Enterprise Manager service.
4. Run the following command to set the repository password:

```
aemctl.exe repository setpassword --master-user-password your_MasterUserPassword --  
repository-password your_RepositoryPassword
```

Example:

```
aemctl.exe repository setpassword --master-user-password  
ANqagYERP3UKmGLK6UNuMqrkAGxwH8FM --repository-password 1234567890123456789000
```

Steps 1-4 only need to be performed the first time you want to encrypt the repository file. If you subsequently need to decrypt the repository file and then re-encrypt it, they are not required.

5. Run the following command to encrypt the repository:

```
aemctl.exe repository secure --on --master-user-password your_MasterUserPassword
```

Example:

```
aemctl.exe repository secure --on --master-user-password  
ANqagYERP3UKmGLK6UNuMqrkAGxwH8FM
```

6. Restart the Qlik Enterprise Manager service.

To disable encryption for the repository:

- Run the following command:

```
aemctl.exe repository secure --off --master-user-password your_MasterUserPassword
```

For information on setting user permission, see [User permissions \(page 298\)](#).

For more information on commands related to the master user password, see [Commands Related to the Master User Password](#).

5.7 Controlling execution of user-defined commands

Several endpoints support running user-defined commands for pre/post-processing files. In the [Operating System Level Credentials](#) tab shown in SERVER view, you can provide user credentials for user-defined commands that needs to be executed at operating system level. By default, such commands are executed under the Replicate Server service account. This may constitute a security risk, as it allows any Replicate user with Admin or Designer permissions to specify user-defined commands that could lead to a full compromise of the server.

Note that when Replicate Server is installed on Linux, the [External utilities \(page 58\)](#) tab will be hidden as the potential security risk is relevant to Windows only.

You can also determine whether to allow user-defined commands to be executed at all.

To do this:

1. On the Replicate Server machine, open the `<PRODUCT_DIR>\bin\repctl.cfg` file and set the `enable_execute_user_defined_commands` parameter to "true" or "false" (the default) as required.
2. If **Another account** is selected in the [Operating System Level Credentials](#) tab and you changed the default Attunity Replicate Server service Log On user (see the Replicate Help for instructions on how to do this):

- a. For both admin and non-admin users:
Add the user specified in the Attunity Replicate Server service **Log On** tab to the "Replace a process level token" policy in the "Local Security Policy" settings.
- b. For non-admin users only:
 - i. Grant the user full control over the Replicate **data** folder.
The default location of the **data** folder is *C:\Program Files\Qlik\Replicate\data*
 - ii. Grant the user full control over the folder containing the user-defined command to be executed.



*The user must be the same as the non-admin user specified in the Attunity Replicate Server service **Log On** tab.*

3. Restart the Qlik Replicate services.

Executing operating system commands as a different user

The load utility of the following target endpoints can run via user-provided credentials instead of the LocalSystem user:

- Google Cloud BigQuery
- Netezza Performance Server
- PostgreSQL-based

6 Managing servers

Servers View is where you manage the Replicate and Compose Servers to be monitored by Enterprise Manager. This view also provides information about the number of tasks defined on each server as well as their current status. To switch from **Tasks** View to **Servers** View, click the **Servers** tab in the top left of the console.



Some of the management options described in this chapter are available for Replicate servers only. In such cases, a note will clearly indicate that the described management option is not available for Compose servers.

In this section:

- [Server requirements \(page 49\)](#)
- [Adding Servers \(page 50\)](#)
- [Monitoring servers \(page 52\)](#)
- [Server settings \(page 54\)](#)
- [Additional server management options \(page 64\)](#)
- [Registering a license for a monitored server \(page 66\)](#)
- [Viewing server messages \(page 68\)](#)

6.1 Server requirements

The following section lists the requirements for connecting to a Replicate or Compose Server.

Qlik Replicate Server requirements

To be able to connect to a Replicate Server, Enterprise Manager requires the following for each Replicate Server to be monitored:

- A Replicate administrator user. Enterprise Manager uses this user to connect to the Qlik Replicate instance. This means:
 - If the connection is made directly to the replication server, you can use the fixed user name `admin`. In this case, the default port is 3552. You set the `admin` user password on the Replicate server using the `repctl setserverpassword` command, as follows:
On the Replicate server, run the following command from the `bin` directory; then restart the Replicate service:
`repctl.exe setserverpassword <adminPassword>`
For more details, see the Replicate documentation.
 - If the connection is made to the Replicate UI server, you must specify a user with Admin role. This can be the user that installed Replicate, an Active Directory user with the Replicate Admin role, or a member of the `AttunityReplicateAdmins` Active Directory group. The default port is 443. For information on setting user roles within Qlik Replicate, refer to the *Qlik Replicate Setup and User Guide*.

It is recommended that you configure this user with a strong password that does not need to be changed frequently. If the administrator password changes, you also need to change it in Enterprise Manager.



If you need to change the password and the associated user is an Active Directory user, you must stop monitoring all servers with the same user before changing the password in Active Directory.

- The port that Enterprise Manager uses for connecting to Qlik Replicate needs to be opened inbound on the Replicate server machine. Depending on your network architecture (for example a WAN environment), you may also need to open ports on routers and border firewalls. If you block outbound traffic from the Enterprise Manager machine, you need to make an exception on the Enterprise Manager machine for the ports used to connect to the Replicate Server.

For additional information, see [Software requirements \(page 16\)](#).

Qlik Compose Server requirements

To be able to connect to a Compose Server, Enterprise Manager requires the following for each Compose Server to be monitored:

- A Compose administrator user. Enterprise Manager uses this user to connect to the Qlik Compose instance.
It is recommended that you configure this user with a strong password that does not need to be changed frequently. If the administrator password changes, you also need to change it in Enterprise Manager
- The port that Enterprise Manager uses for connecting to Qlik Compose needs to be opened inbound on the Compose server machine. Depending on your network architecture (for example a WAN environment), you may also need to open ports on routers and border firewalls. If you block outbound traffic from the Enterprise Manager machine, you need to make an exception on the Enterprise Manager machine for the ports used to connect to the Compose Server.

6.2 Adding Servers

You can add both Replicate Servers and Compose Servers to Enterprise Manager. Once a Server is added and monitoring is enabled, Enterprise Manager establishes a connection to it and automatically discovers and retrieves all tasks and messages, along with basic information about the server.



You need the admin role to be able to add a server.

To add a server:

1. Open Qlik Enterprise Manager.
2. Click **Add Server**.
The **Add Server** dialog box opens.

3. Provide the following information:

- **Name:** A unique name not exceeding 64 characters. The name can contain Latin characters, spaces, dots, dashes, and underscores.
- **Description:** Optional. Cannot exceed 250 characters.
- **Type:** Choose **Replicate** or **Compose** according to the server you wish to add.
- **Host:** The server's host name or IP address. The combination of host and port must be unique.
- **Port:** The port on which the host listens. The combination of host and port must be unique.
- **Username:** The user name for the server. Enterprise Manager uses this user to connect to the server. The name can contain Latin characters, spaces, dots, dashes, and underscores. For a domain user, use the following format: <domain>\<user name>
For example: Qlik_LOCAL\JohnMiller



The user must be a user with admin privileges. For more information, see [Server requirements \(page 49\)](#).

- **Password:** The user password for the server.



If you need to change the password and the associated user is an Active Directory user, you must stop monitoring all servers with the same user before changing the password in Active Directory.

For details, see [Server requirements \(page 49\)](#).

4. Verify server certificate.

Select this option to ensure the Server certificate is trusted. As a rule, to reduce the chance of "man-in-the-middle" attacks, this option should always be selected.

- When connecting directly to an Replicate replication server (default port 3552) with its automatically generated self-signed certificate, Enterprise Manager is able to validate the certificate without requiring any additional setup.
- When connecting to a Replicate Server via the Replicate UI Server (typically using port 443) or to the Replicate replication server with a user-installed certificate, you must make sure that the SSL/TLS certificate used by the server is trusted by the Enterprise Manager machine. The same applies when connecting to a Compose Server with a user-installed certificate. You can easily verify whether the certificate is trusted by opening a Chrome browser window on the Enterprise Manager machine and connecting to Replicate. If there are no security warnings, the certificate is trusted.

For information on the different ways of connecting to Qlik Replicate, see [Qlik Replicate Server requirements \(page 49\)](#).

5. Select the **Monitor this server's tasks and messages** check box if you want Enterprise Manager to retrieve tasks and messages from this server.
6. Click **Test Connection** to verify that Qlik Enterprise Manager is able to establish a connection to the server.
7. Click **OK** to save your settings.

6.3 Monitoring servers

Qlik Enterprise Manager shows server information in table format. The following columns are available. Note that columns marked with an asterisk (*) are *not* available for Compose servers.

General columns:

- **State:** The current state of the server. When the server is being monitored, the state can be **Monitored** (green icon), which means that the Enterprise Manager connected and synchronized successfully, or **Error**, including error details (red icon). When the server is not being monitored, the state is **Not Monitored** and the connection icon is grayed out.



*When a server experiences connection issues, tasks for this server are grayed out in the **Tasks View**.*

- **Name:** The server's display name
- **Host:** The server's hostname or IP address
- **Port:** The port through which the server is accessed
- **Type:** The server version (i.e. Compose or Replicate)
- **Version:** The server version
- **License Expiration:** The expiration date of the server license
- **Last Connection:** The date and time of the last successful sync/retrieval of tasks and messages
- **Message:** The error message displayed if Qlik Enterprise Manager cannot connect to the server

Not displayed by default:

- **Days to Expiration:** Days remaining until the license expires
- **Description:** A description of the server
- **Issue Date:** When the license was issued
- **License State:** The current license state (e.g. valid, expired, etc.)
- **Platform:** The operating system on which the server is installed, which will be Windows or Linux for Replicate servers or Windows for Compose servers.

*Resource utilization columns:

- ***Disk Usage (MB):** The amount of disk space that the server is currently consuming. This is the sum of disk usage of all tasks on this server. For details about a task's disk usage, see [Task progress summary \(page 221\)](#).
- ***Memory (MB):** The amount of virtual memory that the server is currently consuming. This is the sum of virtual memory usage of all active tasks on this server, not including stopped tasks. For details about a task's memory usage, see [Task progress summary \(page 221\)](#).

Not displayed by default:

- ***Qlik CPU (%)**: The amount of CPU being utilized on the server machine by Replicate and Enterprise Manager processes only.
- ***Machine CPU (%)**: The amount of CPU being utilized by *all* processes on the server machine (including Replicate and Enterprise Manager processes).

Tasks columns: The number of tasks for each state.

Task state columns

Task State	Icon	Description
Total	N/A	The total number of tasks, regardless of state.
Running		The number of running tasks
Stopped		The number of stopped tasks
Recovering		The number of recovering tasks.
Error		The number of tasks that encountered a fatal error

Customizing server columns

You can use the **Columns Settings** dialog box to select the columns to display and to arrange the order in which they appear. In addition, from the context menu, you can hide a column, export the list to a TSV file, or sort by column in ascending or descending order. For more information, see [Customizing task columns \(page 249\)](#).

Searching for servers

You can search for specific servers by typing a string into the **Search Servers** box above the table. Note that Enterprise Manager searches only textual columns, not numeric columns. The following columns are included in the search, even if a column is not displayed in the user interface:

- State
- Name
- Host
- Port
- Description
- License Expiration
- Message
- Platform
- Version

You can also restrict a search to a specific column by preceding the search string with the column name and a colon, as follows: `columnName:value` (for example: `name:john-vm.abc.local`). This is applicable to all available columns, not only the columns listed above.



When searching for a number, only enter whole numbers, no commas or special characters. For example, to search for 2,500, as displayed in the user interface, enter 2500; to search for 100%, enter 100.

6.4 Server settings

You can modify Replicate and Compose server settings.



This feature requires the monitored Replicate servers to be version 6.0 or later. Currently, only the logging settings for Compose servers can be modified.

To modify server settings, switch to **Servers** view and select the desired server. Then, from the **Server Management** drop-down menu, select **Settings**.

The **Settings for Server "<Name>"** window opens.

The following items are displayed on the left of the window:

- [Global error handling \(page 54\)](#) (Replicate servers only)
- [Resource Control \(page 55\)](#) (Replicate servers only)
- [File Transfer Service \(page 56\)](#) (Replicate servers only)
- [External utilities \(page 58\)](#)
- [Logging \(page 59\)](#)
- [More options \(page 220\)](#)

Global error handling

You can configure how Qlik Replicate responds to specific types of errors. You can define error handling on the task level or the server level. The configurations you make in the Server Settings affect all tasks created for this instance of Qlik Replicate unless you define a task to use the definitions you create for that task. For information on how to configure error handling for a specific task, see [Error handling \(page 203\)](#) in the [Customizing tasks \(page 91\)](#) chapter.



- Changes to settings will only take affect after restarting all tasks.
- Global error handling is not available when the [Apply changes using SQL MERGE](#) task setting is enabled.

The following sub-tabs are available:

- **Environmental Errors:** An error that is caused by an environmental problem in the source or target endpoint or on the network. Environmental errors can be restarted.

The information you enter in this tab is the same as the information you enter in the **Environmental Errors** tab for tasks. For information about the options available in this tab, see [Environmental errors \(page 204\)](#).

- **Data Error:** An error related to data processing at the record level.
The information you enter in this tab is the same as the information you enter in the **Data Error** tab for tasks. For information about the options available in this tab, see [Data Errors \(page 205\)](#) in the Customizing Tasks chapter.
- **Table Error:** An error in processing data or metadata for a specific table. This only includes general table data and not an error that relates to a specific record.
The information you enter in this tab is the same as the information you enter in the **Table Error** tab for tasks. For information about the options available in this tab, see [Table Errors \(page 208\)](#) in the Customizing Tasks chapter.
- **Apply Conflicts:** Errors that occur when the target endpoint is not synchronized with the source endpoint when processing changes. This can cause duplicate key errors on INSERT operations or zero rows affected on UPDATE/DELETE operations.
The information you enter in this tab is the same as the information you enter in the **Apply Conflicts** tab for tasks. For information about the options available in this tab, see [Apply Conflicts \(page 208\)](#) in the Customizing Tasks chapter.

Resource Control

You can set high and critical disk space and memory utilization thresholds.



Disk space is checked only for the drive where the data folder resides.

Thresholds are calculated as a percentage of total capacity. So, for example, a disk space utilization threshold of 80% would mean that 20% of available disk space remains.

After setting the thresholds, you can click the **New Notification** button to [define a notification](#) that will be sent whenever a given threshold is exceeded and/or returns to normal.

Disk space control

Before you can modify settings, you first need to select the **Disk Space** check box. To disable Disk Space resource control, clear the **Disk Space** check box.

In the **High Disk Space Utilization Threshold** section, specify the high disk space utilization threshold (in terms of percentage). When the threshold is reached, a notification will be sent (if defined).

In the **Critical Disk Space Utilization Threshold** section, specify the critical disk space utilization threshold (in terms of percentage). When the threshold is reached, all tasks will be stopped and a notification will be sent (if enabled). Replicate will resume the tasks automatically when there is sufficient disk space to do so.

System memory control

Before you can modify settings, you first need to select the **System Memory** check box. To disable System Memory resource control, clear the **System Memory** check box.

Memory utilization is calculated using the following formula (note that “swap file” is used generically to refer to both page file memory on Windows and swap file memory on Linux):

$$\frac{(\text{used_swap_file} + \text{used_physical_memory})}{(\text{total_swap_file} + \text{total_physical_memory})} * 100$$

Example:

$$(5 \text{ GB} + 5 \text{ GB}) / (10 \text{ GB} + 10 \text{ GB}) * 100 = 50\%$$

In the **High System Memory Utilization Threshold** section, specify the high system memory utilization threshold (in terms of percentage). When the threshold is reached, a notification will be sent (if defined).

In the **Critical System Memory Utilization Threshold** section, specify the critical system memory utilization threshold (in terms of percentage). When the threshold is reached, Replicate will start stopping tasks and a notification will be sent (if enabled). The tasks will be resumed automatically when there is sufficient memory to do so.

File Transfer Service

The Qlik File Transfer Service (FTS) is a robust and reliable file transfer engine designed to efficiently transfer files over the WAN. This can dramatically improve transfer speeds when the source endpoint and the target endpoint are located on different LANs.



Changes to settings will only take affect after restarting all tasks.

How it works

A solution using FTS consists of two Qlik Replicate Servers: A local Qlik Replicate Server installed on the source endpoint LAN and a remote Qlik Replicate Server installed on the target endpoint LAN.

A local task on the local server is defined from the source endpoint to a File Channel target. A remote task on the remote Qlik Replicate Server is defined from a File Channel source to the target endpoint.

The FTS runs on the remote Qlik Replicate Server only and transfers the File Channel files from the storage location defined in the local task to the storage location defined in the remote task.

Upon file transfer, and before [Compression \(page 56\)](#) and [Encryption \(page 57\)](#), large files are split into smaller blocks which form recoverable transport units, and small files are merged into bigger blocks to be sent at the same time. The blocks are then transferred and reconstructed into File Channel files when received by the FTS server.

For information on setting up a File Channel source or target to use FTS, refer to the *Qlik Replicate Setup and User Guide*.

Compression

File Channel files are compressed upon sending using GZIP. You can disable the compression and control the compression level.

Encryption

After compression, File Channel files are encrypted using a randomly generated AES-256 session key. The session key is exchanged between the client and server using the Diffie-Hellman key exchange protocol which is authenticated using a secret key that is shared between the client and the server.



The File Transfer Service should be configured on the remote Qlik Replicate Server only.

Defining a File Transfer Service

Define a File Transfer Service as described below.

To add a File Transfer Service:

1. In the **File Transfer Service** tab, click **Add File Transfer Service**. The **Add File Transfer Service** window opens.
2. Edit the values in the **Name**, **Host** and **Port** columns as follows:
 - **Name:** The name of the File Transfer Service.
 - **Host:** The host name or IP address of machine on which the remote Qlik Replicate Server is installed. The default is 0.0.0.0 (all interfaces). If the server has multiple NICs (Network Interface Cards), you can define a different File Transfer Service for each card.
 - **Port:** The port through which the File Channel files are received.
 - **Enabled:** select the check box to enable the File Transfer Service.
3. Click **Save** to save your settings.

Editing a File Transfer Service

You can edit a File Transfer Service as described below.

To edit a File Transfer Service:

1. Select the File Transfer Service you want to edit.
2. Edit the values in the **Name**, **Host** and **Port** columns as follows:
 - a. Click the cell to make it editable.
 - b. Change the value as required and then click **Save**.



When you edit a File Transfer Service, make sure that any File Channel targets configured to use the File Transfer Service are also updated accordingly. For more information on File Channel Targets, refer to the Qlik Replicate Setup and User Guide.

Deleting a File Transfer Service

You can delete File Transfer Services that you no longer want to use.

To delete a File Transfer Service:

1. In the File Transfer Services List, select the item you want to delete.
2. Click the **Remove** button.

External utilities

Several endpoints support running user-defined commands for pre/post-processing files. In this tab, you can provide user credentials for user-defined commands that needs to be executed at operating system level.

By default, such commands are executed under the Replicate Server service account. This may constitute a security risk, as it allows any Replicate user with Admin or Designer permissions to specify user-defined commands that could lead to a full compromise of the server.

Note that when Replicate Server is installed on Linux, this tab will be hidden as the potential security risk is relevant to Windows only.



Changes to settings will only take affect after restarting all tasks.

Endpoints that support user-defined commands are as follows:

- Amazon S3 target
- File source
- File target
- Google Cloud Storage target
- Microsoft Azure ADLS target



IMPORTANT Execution of user-defined commands is blocked by default. To allow execution of user-defined commands, follow the procedure described in [Controlling execution of user-defined commands \(page 47\)](#).

To run user-defined commands under the Replicate service (Log On) account (the default):

- Select **Replicate service account**.

To run user-defined commands under a different account:

1. Select **Another account** and then specify a user name and password in the designated fields.
For domain users, the user name should be entered in the following format: `domain/Username`
For local users, the `domain` is not required.
2. If you changed the default Attunity Replicate Server service Log On user, add the user specified in the Attunity Replicate Server service **Log On** tab to the "Replace a process level token" policy in the "Local Security Policy" settings.

Logging

The following topics describe the log management options:

- [Setting Logging Levels for the Server and File Transfer Service](#)
- [Setting Automatic Roll Over and Cleanup](#)
- [Viewing and downloading logs \(page 61\)](#)
- [Deleting server, task and FTS log files \(page 62\)](#)

Setting logging levels

You set the logging level for the Replicate Server, File Transfer Service logs, Compose Server and Compose Agent logs in **Server** view. The level you set determines what information is written to the logs. The Server logs provide information about the Qlik Replicate Server instance you are working with as opposed to individual tasks. For information on configuring the task logs, see [Logging \(page 210\)](#).

The following logging levels are available, ordered from the lowest level to the highest:

1. Errors
2. Warnings
3. Info
4. Trace
5. Verbose

The higher levels always include the messages from the lower levels. Therefore, if you select **Error**, only error messages are written to the log. However, if you select **Info**, informational messages, warnings, and error messages are included. Selecting **Verbose** writes all possible messages to the log.

You can set a global logging level for all components or you can set a separate logging level for each component.

To set the logging levels:

1. To set a global logging level, move the top slider (the slider with the labels) to the log level you want. Note that all of the sliders for the individual modules move to the same position that you set in the main slider.
2. Make any changes to the sliders for the individual modules. This is optional. Note that if you change the main slider, all of the individual sliders are reset to the new position. If you want to maintain a different logging level for a specific module, you need to reset it.
3. Click **Save** at the bottom of the window.

Storing trace and verbose logging in memory



This option is relevant for Replicate tasks only and is not supported with the following source endpoints:

- *MongoDB*
- *Salesforce (Streaming CDC)*
- *Salesforce (Incremental Load)*
- *SAP Application*
- *SAP ODP*

When the logging level is set to "Trace" or "Verbose", you can instruct Replicate to store the logging information in memory until an error occurs. On detecting an error, Replicate will begin writing to the physical logs and continue to do so for a few minutes after the initial occurrence of the error.

If *no* error occurs before the allocated memory is used up, Replicate will empty the memory buffer and start afresh.

This option is useful for tasks that fail unpredictably and for no obvious reason. The problem with continually writing large amounts of information to the logs is twofold:

- Running in "Trace" or "Verbose" logging mode will quickly use up available disk space (unless the logging settings have been configured to prevent this).
- Continually writing large amounts of data to the logs will affect performance.

To use this option

1. Select the **Store trace/verbose logging in memory, but if an error occurs, write to the logs** check box at the top of the tab.
2. In the **Allocate memory up to (MB)** field, specify the amount of memory you want to allocate for storing logging information.

Setting automatic roll over and cleanup

In the **Log File Management** tab, you can define when to roll over the log files and when to delete old log files from the system.

Automatic rollover

You can determine when to stop logging to the current log file and begin to log to a new log file. Rolled over log files are appended with a 12-digit timestamp.

- **Roll over the log if the log file is older than (days):** Select the check box and then specify the maximum number of days the current log file is allowed to exist before being rolled over. The default value is **7** days.



This option is not available for Compose servers.

- **Roll over the log if the log file is larger than (MB):** Select the check box and then specify the maximum number of megabytes the current log file is allowed to reach before being rolled over.



When the **Store trace/verbose logging in memory, but if an error occurs, write to the logs** option is enabled, the actual size of the `repsrv.log` may reach the sum of the **Allocate memory up to (MB)** size and the **Roll over the log if the log file is larger than (MB)** size, before it is rolled over.

For more information on the "Store trace/verbose logging in memory" option, see [Setting logging levels \(page 59\)](#)

The default value is **100** megabytes.



- If you edit this setting while tasks are running, the new setting will not affect the task log files until the tasks are stopped and then resumed. The server log files are not affected by this limitation.
- The scheduled process (`LogFileCleanLogs`) that checks the log file size runs every five minutes. Consequently, the actual size/age of the rolled over log file may deviate slightly from the specified value(s).
- The time of a file is determined by its "modified" time as opposed to its "created" time.

Automatic cleanup

You can determine the maximum number of days old log files (i.e. log files that have been rolled over) are retained before being deleted.

- **Delete log files that are older than (days):** Select the check box and then specify the maximum number of days to retain a saved log file. Log files that are older than the specified number of days will be automatically deleted from the system. For example, if you specify **4**, then on the fifth day, any log file older than **4** days will be deleted.

The default value is **45** days.

Viewing and downloading logs

You can view the server or File Transfer Service log files and download them if necessary.



File Transfer Service log files are not relevant for Compose servers.

To view or download log files:

1. In **Servers** view:
 1. Select a server and then select **Settings** from the **Server Management** drop-down menu.
 2. Select **Logging** on the left of the **Settings for Server <server_name>** window.
2. Select the **Server Logging Levels** or the **File Transfer Service Logging Level** tab as required.

3. Click the **View Logs** button.
The **Log Viewer** window opens.
4. Continue from step 2 in [Viewing and downloading log files](#).

Deleting server, task and FTS log files

You can manually delete task, server, and File Transfer Service log files older than the specified number of days.



Log files from Compose servers cannot be deleted.

To delete the log files:

1. In **Servers** view:
 1. Select a server and then select **Settings** from the **Server Management** drop-down menu.
 2. Select **Logging** on the left of the **Settings for Server <server_name>** window.
2. Click the **Delete Logs** button in any of the available tabs.
The **Delete Logs** window opens.
3. Select which logs to delete and, for each log, optionally change the default number of days (45).
4. Click **Delete**.
Selected logs older than the specified number of days will be deleted.

More options

These options are not exposed in the UI as they are only relevant to specific versions or environments. Consequently, do not set these options unless explicitly instructed to do so by Qlik Support or product documentation.

To set an option, simply copy the option into the **Add feature name** field and click **Add**. Then set the value or enable the option according to the instructions you received.

Server management permissions

The following table describes the required permissions for viewing and managing server settings.



Only permissions related to logging settings are relevant for Compose servers.

For information on managing user permissions, see [Managing user permissions \(page 307\)](#).

Global Error Handling setting permissions

Permission To	Requires: Admin	Requires: Designer	Requires: Operator	Requires: Viewer
View Global Error Handling	All Tasks	All Tasks	-	-
Edit Global Error Handling	All Tasks	All Tasks	-	-

Server Logging Levels setting permissions

Permission To	Requires: Admin	Requires: Designer	Requires: Operator	Requires: Viewer
View server logging levels	Server	Server	Server	-
Edit server logging levels	Server	Server	Server	-
View/download server logs	Server	Server	Server	-
Delete server/FTS logs	Server	Server	Server	-
Delete task logs	Server	Server	Server	-

FTS Logging Levels setting permissions

Permission To	Requires: Admin	Requires: Designer	Requires: Operator	Requires: Viewer
View FTS log levels	Server	Server	Server	-
Edit FTS log levels	Server	Server	Server	-
View/download FTS logs	Server	Server	Server	-
Delete server/FTS logs	Server	Server	Server	-
Delete task logs	Server	Server	Server	-

Compose Agent Logging setting permissions

Permission To	Requires: Admin	Requires: Designer	Requires: Operator	Requires: Viewer
View Compose Agent log levels	Server	Server	Server	-
Edit Compose Agent log levels	Server	Server	Server	-
View/download Compose Agent logs	Server	Server	Server	-

Log File Management setting permissions

Permission To	Requires: Admin	Requires: Designer	Requires: Operator	Requires: Viewer
View Cleanup/Rollover policy for server/FTS/Compose Agent	Server	Server	Server	-
View Cleanup/Rollover policy for tasks	Server	Server	Server	-
Edit Cleanup/Rollover policy for server/FTS/Compose Agent	Server	Server	Server	-
Edit Cleanup/Rollover policy for tasks	Server	Server	Server	-
Delete server/FTS logs	Server	Server	Server	-
Delete task logs	Server	Server	Server	-

Resource Control setting permissions

Permission To	Requires: Admin	Requires: Designer	Requires: Operator	Requires: Viewer
View Resource Control	Server	Server	Server	-
Edit Resource Control	Server	Server	-	-

File Transfer Service (FTS) setting permissions

Permission To	Requires: Admin	Requires: Designer	Requires: Operator	Requires: Viewer
View FTS list	Server	Server	Server	-
Add/Edit/Remove FTS	Server	Server	-	-

6.5 Additional server management options

In addition to adding servers, you can also perform other management tasks, as described in the following table. Note that before performing any of the actions, you must first select the desired server or servers (multiple selection is supported for some actions).

Server management actions

To	Do this
Edit a server	<p>Double-click the server.</p> <p>OR</p> <p>Right-click the server and select Connection Properties.</p> <p>OR</p> <p>Select the server and then click the Connection Properties button on the toolbar.</p> <p>Edit the information as described in Adding Servers (page 50).</p>
Manage Endpoints Connections (Replicate Servers only)	<p>Click Manage Endpoints.</p> <p>Edit the information as described in Adding an Endpoint.</p>
Create a new task on the server (Replicate Servers only)	<p>Click New Task.</p> <p>Edit the information as described in Setting up Tasks.</p>
View server log files	<p>If you need more detailed information about a server, you can view the log files directly from within Enterprise Manager. See Viewing and downloading log files (page 286).</p>

To	Do this
<p>Start monitoring or stop monitoring a server</p> <p>Stopping monitoring servers may be useful when a maintenance window, upgrade, or known outage is coming up, or when you want to focus on a single server or a set of servers without being disrupted by messages and tasks from other servers.</p> <div style="border: 1px solid #ccc; padding: 5px; margin: 10px 0;"> <p> <i>When you stop monitoring a server, the server appears as disabled in the Servers View, the Tasks View does not include any tasks for the respective server, and the Message Center does not show new messages for the respective server.</i></p> </div> <div style="border: 1px solid #ccc; padding: 5px; margin: 10px 0;"> <p> <i>You need the admin role to be able to start or stop monitoring a server.</i></p> </div>	<p>Right-click the server and select Stop Monitoring/Start Monitoring accordingly.</p> <p>OR</p> <p>Select the server and then click the Stop Monitoring or Start Monitoring button on the toolbar.</p>
<p>Delete a server</p> <div style="border: 1px solid #ccc; padding: 5px; margin: 10px 0;"> <p> <i>Deleting a server deletes the tasks and the message history for the respective server.</i></p> </div> <div style="border: 1px solid #ccc; padding: 5px; margin: 10px 0;"> <p> <i>You need the admin role to be able to delete a server.</i></p> </div>	<p>Right-click the server and select Delete.</p> <p>OR</p> <p>Select the server and then click Delete.</p> <p>When prompted to confirm the deletion, click Yes.</p>
<p>Register a server license</p>	<p>See Registering a license for a monitored server (page 66).</p>
<p>Edit user permissions on the server level, for all tasks on the server, or for all endpoints (relevant to Replicate Servers only) on the server.</p>	<p>Click Server Permissions. Follow the instructions in Managing user permissions (page 307).</p>

6.6 Registering a license for a monitored server

You can update an existing Replicate or Compose server license or register a new license directly from Qlik Enterprise Manager.

To register a license:

1. Copy the license file to your computer or any computer in your network you have access to.
2. Select **Register License** from the **Server Management** drop-down menu or right-click the server and select **Server Management > Register License**.
The **Register License** dialog box opens.
3. Do one of the following:
 - Click **Load** and browse to locate and select the license file.
 - Copy the license text and paste it into the **License text** field.

The license text is displayed in the dialog box as shown below. Check to be sure that the details are correct.

Register License... ✕

Load license from file or copy & paste to screen

Load...

License text:

```
#
# Attunity License
# Generated on 24-Feb-2016 11:55:51.6252+02:00
# License Comment:
#
license_type=evaluation_license
licensed_to=Charlie
licensed_by=Attunity Israel
serial_no=60004468
expiration_date=2016-03-25
hosts=
source_types=
target_types=
features=manager
version=5.0
issue_date=2016-02-24
checksum=E7JKC-F3CCQ-EJJ32-22JJC
```

To register new or modified license, click 'Register License'.

Register License Close

4. Click **Register License** to register the license.
A message indicating that the license was registered successfully is displayed.

6.7 Viewing server messages

You can view server messages in the Message Center. The Message Center is located at the bottom of the console and contains messages about the servers and tasks. For more information, see [Messages and notifications \(page 269\)](#).

7 Defining and managing tasks

This section describes how to design a replication task. To design a replication task, you must first be sure that you have configured at least one source endpoint and one target endpoint to work with Qlik Replicate.



The operations described in this section are relevant for Replicate tasks only.



A number of variables affect the amount of tasks that can be run on a single Replicate Server, including the task configuration (e.g. how many tables are being replicated), the size of the source tables and the hardware configuration of the Replicate Server machine. Bearing this in mind, the number of tasks that can be run on a single Replicate Server should not exceed 100 (and may need to be significantly less depending on the aforementioned variables). Best practice is to perform load testing in a Test environment before moving to Production.

In this section:

- [Adding tasks \(page 69\)](#)
- [Adding a source and target endpoint to a task \(page 74\)](#)
- [Selecting tables and/or views for replication \(page 76\)](#)
- [Editing a replication task \(page 82\)](#)
- [Deleting a replication task \(page 83\)](#)
- [Searching for tasks \(page 82\)](#)
- [Exporting and importing tasks \(page 83\)](#)

7.1 Adding tasks

Before you get started with designing the features that you need for a task, you must first define the task's default behavior.

To add a task:

1. In Tasks view or Servers view, click **New Task**.
The **New Task** dialog opens.

New Task ✕

Server Name

Name ⁱ

Description

Replication Profile
Select the profile that best describes your replication use case

Unidirectional

Bidirectional ⁱ

Log Stream ⁱ

Task Options
Select the replication options for this task



Full Load



Apply Changes



Store Changes

Perform a full load of target tables, then keep them up-to-date by applying changes.

OK

2. From the **Server Name** list, select the server for the task.
You can only select the server name if you open the dialog from the Tasks view. When you open the dialog from the Servers view, the selected server is displayed.
3. Enter a name for the task. The name should be descriptive to indicate the purpose of the task. The name cannot exceed 32 characters, contain non-Latin characters, or contain any of the following characters: | \ / : * ? " < >
4. Optionally, enter a description for the task.
5. Choose one of the following replication profiles:
 - **Unidirectional** - Choose to replicate between endpoints for the purpose of Unidirectional.
 - **Bidirectional** - Choose to synchronize records between two endpoints.

For more information, see the instructions on setting up [Bidirectional replication \(page 71\)](#).

- **Log Stream Staging** - Log Stream Staging enables a dedicated Replicate task to save data changes from the transaction log of a single source database and apply them to multiple targets, without the overhead of reading the logs for each target separately.

For information about Log Stream Staging, refer to the [Qlik Replicate online help](#).

6. Select task options:

- **Full Load**: Click to enable or disable Full Load options for this task.
When full load is enabled, Enterprise Manager loads the initial source data to the target endpoint. By default, a full load is carried out for this task. If you want to change this setting after you begin working with this task, you make the change in the Task Settings, [Full Load \(page 187\)](#) tab.
- **Apply Changes**: Click to enable or disable Apply Changes (Change Processing).
When this option is enabled, Enterprise Manager keeps the target tables continually up-to-date with any changes made to the source tables. By default, change processing is carried out for this task. You can view the change processing in the Monitor view.
For more information, see [Monitoring Change Processing replication \(page 233\)](#). If you want to change this setting after you begin working with this task, you make the change in the Task Settings, [Change processing \(page 190\)](#) tab.



When the **Bidirectional** replication profile is selected, the **Apply Changes** option cannot be disabled.

- **Store Changes**: Click this button to enable or disable **Store Changes**.
If this option is enabled, changes are stored in change tables or in an audit table. By default, changes are not stored.
For information about storing and applying changes, see the [Qlik Replicate online help](#).



When the **Bidirectional** replication profile is selected, the **Store Changes** button will be unavailable.

7. Click **OK** to close the **New Task** dialog box and save your settings.

Bidirectional replication

Bidirectional replication enables organizations to synchronize data between two endpoints (henceforth referred to as Endpoint A and Endpoint B), ensuring that both endpoints contain identical records. The endpoints can either be the same type (e.g. Oracle-to-Oracle) or different types (e.g. Microsoft SQL Server-to-Oracle). To implement bidirectional replication, two **Bidirectional Replication** tasks need to be defined: one that captures changes made to Endpoint A and replicates them to Endpoint B (Task 1) and another that captures changes made to Endpoint B and replicates them to Endpoint A (Task 2). An explanation of how to set up these tasks is provided in the following sections.

Limitations

The following limitations apply to bidirectional replication tasks:

- Bidirectional replication does not currently support conflict resolution. To prevent conflicts, organizations should ensure that the application that updates the endpoints participating in a bidirectional replication task, does not simultaneously update the same record in both endpoints. In other words, if a record in Endpoint A was updated, the equivalent record in Endpoint B should only be updated after the update from Endpoint A is replicated to Endpoint B.
- Bidirectional replication tasks currently only support DDL statements from one of the sources involved in the task only. This means that for one of the sources, you must set the [Apply Changes DDL Handling Policy](#) to "Ignore" for all of the options (DROP, TRUNCATE, and ALTER).



The CREATE TABLE DDL is not supported.

- To ensure that the source and target endpoints are identical, transformations and filters should not be used in bidirectional replication tasks.
- The **Use direct path full load** option in the Oracle target endpoint settings is not supported.
- The **Stopping the Task after Full Load** options in the task settings' [Full Load Settings](#) tab is not supported.
- The task's **Change Processing Mode** must be set to [Transactional apply](#).

Supported endpoints

Bidirectional tasks support the following endpoints:

Source Endpoints:

- Amazon RDS for MySQL
- Amazon RDS for PostgreSQL
- Amazon RDS for SQL Server
- AWS Aurora Cloud for PostgreSQL
- File Channel
- Google Cloud SQL for MySQL
- Google Cloud SQL for SQL Server
- Google Cloud SQL for PostgreSQL
- Google Cloud AlloyDB for PostgreSQL
- IBM DB2 for iSeries
- IBM DB2 for LUW
- IBM DB2 for z/OS
- Microsoft Azure SQL (MS-CDC)
- Microsoft Azure SQL Managed Instance
- Microsoft SQL Server
- Microsoft SQL Server (MS-CDC)
- MySQL
- Oracle
- PostgreSQL
- SAP Sybase ASE

Target Endpoints:

- File Channel
- Google Cloud SQL for MySQL
- Google Cloud SQL for PostgreSQL
- Google Cloud AlloyDB for PostgreSQL
- IBM DB2 for z/OS
- Microsoft SQL Server
- Microsoft Azure SQL Database
- MySQL
- ODBC
- Oracle
- PostgreSQL
- SAP Sybase ASE

Setting up Bidirectional replication

This section describes how to set up a Bidirectional replication task in Enterprise Manager.

To set up Bidirectional Task 1:

1. Define a **Bidirectional Replication** task that replicates data from Endpoint A to Endpoint B.



*In a bidirectional replication task, **Full Load** replication is not enabled by default since it is assumed that both endpoints contain identical tables. If this is not the case (for instance, if Endpoint A contains tables that do not exist in Endpoint B), enable **Full Load** replication as well.*

2. Specify a source and target **Loopback prevention table schema** in the task settings' **Loopback Prevention** tab. For more information about loopback prevention settings, see [Bidirectional \(page 186\)](#).
3. Run the task.

To set up Bidirectional Task 2:

1. Define another **Bidirectional Replication** task that replicates data from Endpoint B to Endpoint A.
2. Specify a source and target **Loopback prevention table schema** in the task settings' **Loopback Prevention** tab. For more information about loopback prevention settings, see [Bidirectional \(page 186\)](#).
3. If Full Load was enabled when replicating data from Endpoint A to Endpoint B, you must first wait for the Full Load replication to complete before running the task. Otherwise, run the task immediately.

Using bidirectional replication with the File Channel endpoint

You can use bidirectional replication together with the File Channel endpoint. This is useful if you need to synchronize two endpoints that are either not able to communicate with each other (i.e. are not physically connected) or are located in the WAN. The process involves setting up six separate tasks: Two Full Load-only Unidirectional tasks and four Apply Changes-only Bidirectional tasks.

For information on setting up the File Channel endpoint, see the Qlik Replicate Setup and User Guide.

To set up bidirectional replication with File Channel Endpoints:

1. Set up and run two Full Load only Unidirectional tasks.

Example (FC = File Channel):

Task 1: MySQL --> FC Target **Task 2:** FC Source --> Oracle

2. Wait for the Full Load-only tasks to finish.

3. Set up and run four Apply Changes-only Bidirectional tasks.

Example (FC = File Channel):

Task 1: MySQL Source --> FC Target **Task 2:** FC Source 1 --> Oracle Target

Task 3: Oracle Source --> FC Target 2 **Task 4:** FC Source 2 --> MySQL Target

7.2 Editing and viewing a task description

You can provide a description for specific tasks and then easily view or edit that description as required.

To provide or edit a task description:

1. Open the desired task.
2. In Designer or Monitor view, click the  toolbar button on the left.
3. Enter a description.
4. Click **OK**.

To view a task's description:

1. Open the desired task.
2. In Designer or Monitor view, hover your mouse over the  toolbar button on the left. A tooltip will display the description.

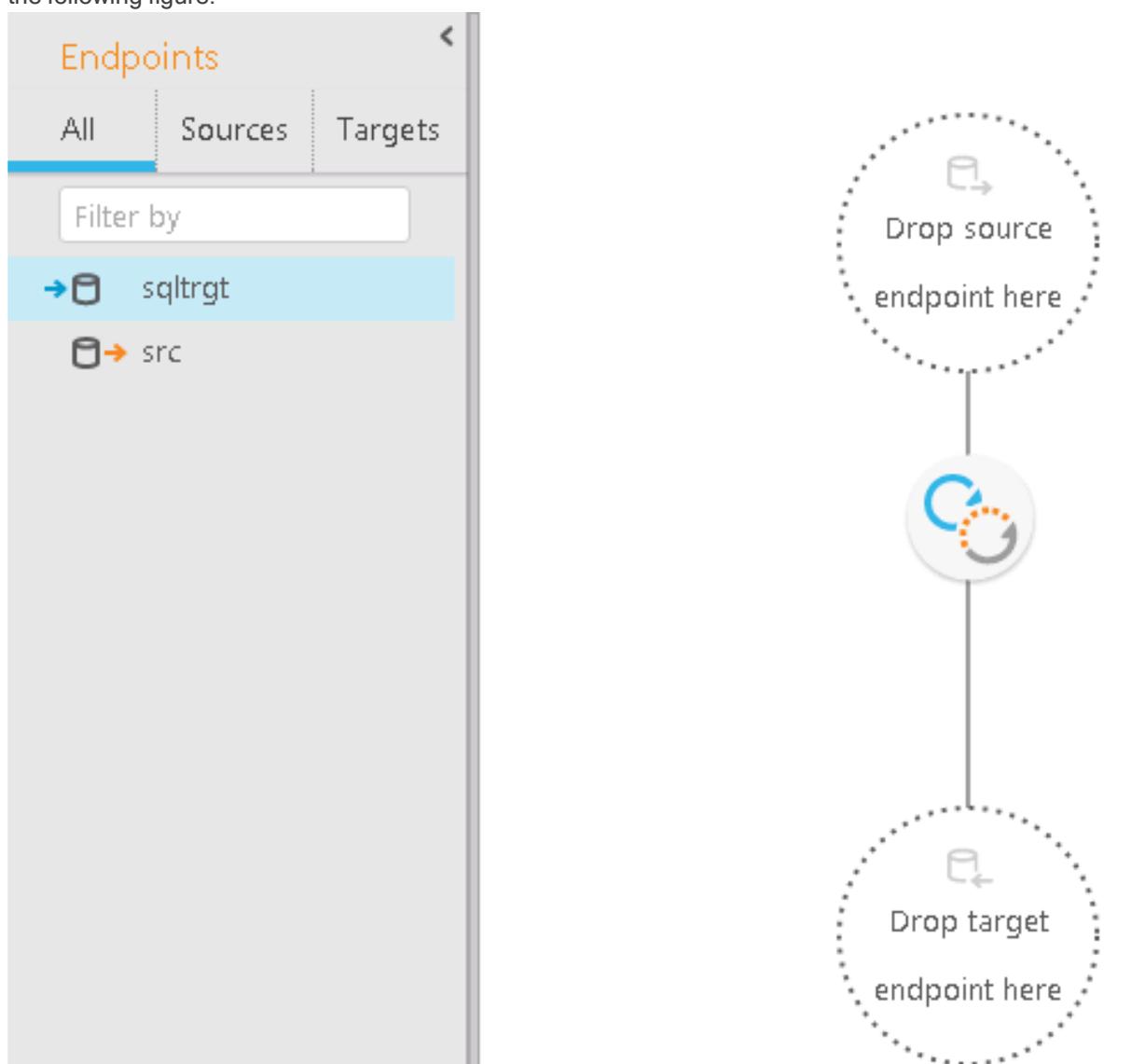
7.3 Adding a source and target endpoint to a task

Once you have added the endpoints, you can design the replication task. The first step in this process is to define the source endpoint where your data is currently stored and the target endpoints where you want to replicate the data. To do this, you just drag one of the endpoints you added into the task map (in Designer mode).

Once you select the endpoint for your task, you must select the tables from the source endpoint to be replicated. The next step in creating a replication task is [Selecting tables and/or views for replication \(page 76\)](#).

To add source and target endpoints to a task:

1. Do one of the following:
 - Create a new task. When you click **OK** in the **Create New Task** dialog box, the task opens on a dedicated tab. For more information, see [Adding tasks \(page 69\)](#).
 - In the Tasks view, select the task to which you want to add endpoints and click **View Task**. The task opens on a dedicated tab.
2. On the dedicated tab for the task, click **Designer** to switch to Designer mode. The Task map is displayed, with the available endpoints listed in the pane on the left, as shown in the following figure.



3. Drag a source endpoint to the top circle in the task map (that contains the text **Drop source endpoint here**). If dragging is not possible, make sure that the endpoint you are using is defined as a source endpoint.
4. Drag a target endpoint to the bottom circle in the task map (that contains the text **Drop target endpoint here**). If dragging is not possible, make sure that the endpoint you are using is defined as a target endpoint.
5. Click **Save**.

7.4 Selecting tables and/or views for replication

This procedure describes how to select the source tables or views that you want to replicate. Note that tables can be selected from any supported endpoint, but views are only supported with the following endpoints:

- Teradata
- PostgreSQL
- MySQL
- SAP Sybase ASE
- IBM DB2 for LUW
- IBM DB2 for z/OS
- Oracle
- Microsoft SQL Server
- ODBC with CDC
- ODBC



- *Replication of views is supported in Full Load Only tasks only, except when replicating from the following sources:*
 - *Teradata*
 - *ODBC with CDC*
- *Views are replicated to the target endpoint as tables*
- *When replicating views, the corresponding tables are created without a primary key. This presents an issue for Apply Changes tasks, which require the target tables to have a primary key. Therefore, if you are also running Apply Changes tasks (using one of the CDC-capable endpoints mentioned above), you need to define one or more primary keys for each of the target tables using a transformation. For an explanation of how to accomplish this, see [Using the Transform tab \(page 95\)](#) in [Defining transformations for a single table/view \(page 93\)](#).*



When working with ODBC with CDC and Teradata source endpoints, any views and tables that you want to replicate must have the same context field(s). If you only want to replicate views, then all of the views must have the same context field(s).

For information on setting up context fields, see "Configuring Change Processing Settings" in the Qlik Replicate Setup and User Guide.

Once you have selected tables/views to replicate, you can run the replication task. However, if you need to make any changes to the structure of the tables in the target endpoint or only select specific columns, you will need to carry out one or both of the following procedures:

To select tables/views:

1. Open the task you are working with if it is not already displayed in a dedicated tab.
For information on opening a task, see [Editing a replication task \(page 82\)](#).
2. In Designer mode, on the right side, click **Table Selection**.
If the source endpoint does not support view selection, the **Select Tables** dialog box opens. If the [source endpoint supports view selection](#), the **Select Tables/Views** dialog box opens.
See the following for information on how to work with the **Select Tables/Select Tables/Views** dialog box:
 - [Searching for tables/views to use in a replication task \(page 77\)](#)
 - [Selecting specific tables/views for replication \(page 78\)](#)
 - [Creating table/view selection patterns \(page 80\)](#)
 - [Setting load order \(page 79\)](#)

Searching for tables/views to use in a replication task

This topic walks you through searching for specific tables/views in preparation for including them in a replication task. You first search for tables that match specific criteria. Then you select the required tables/views from the search results to include them in the task. You can also carry out another search with new criteria and then add additional tables/views to the replication task.

After you finish searching, you can select tables/views for replication. Continue with [Selecting specific tables/views for replication \(page 78\)](#).

To search for tables/views to use in a replication task:

1. In **Designer** mode, click **Table Selection**.
2. In the **Select Tables** dialog box, if the source endpoint supports view selection, select one of the following:
 - **All** to search for tables and views
 - **Tables** to search for tables only
 - **Views** to search for views onlyOtherwise, skip to the next step.
3. From the **Schema** drop-down list, select a table/view schema.

i When working with Microsoft SQL Server, quote characters in schema names - such as "\" or "[\" - must be doubled in order to show the list of tables in the specified schema. For example, **My\\Schema** should be specified as **My\\Schema**.

i When selecting tables from the SAP Application or SAP Application (DB) endpoint, "Business Groups" will appear instead of "Schema".

- Optionally, in the **Table/View** field, type the name or partial name of a table/view you wish to replicate. If you do not do this, Replicate will search for all of the tables/views in the selected schema.

i You can also include special characters in your search string. For more information, see the Note in [Creating a record selection condition for one or more columns \(page 105\)](#).

- Click **Search** to display a list of tables/views.

i When selecting tables from the SAP Application or SAP Application (DB) endpoint, the **Table List** will display all of the tables in the selected Business Group. Hovering your mouse cursor over a table will display a tooltip as shown below.

Table List

FI - FINANCE DOCUM...	AGKO	
FI - FINANCE DOCUM...	BFI	
FI - FINANCE DOCUM...	BFI	

Source: AGKO
Class: Transparent
Description: Cleared Accounts

The **Table List** field displays any table/view that matches the specified search criteria. If the source endpoint supports view selection, an additional **Type** column indicates whether the database object is a table or a view.

- Click **OK**.

See also: [Creating table/view selection patterns \(page 80\)](#).

Selecting specific tables/views for replication

This topic walks you through selecting specific tables/views to replicate.

When you select specific tables/views, all selected tables/views are replicated in full unless you define transformations or filters for the table/view. If you need to make changes to the table/view structures in the target endpoint or if you only want to select specific columns, then you need to perform the procedures described in [Defining transformations for a single table/view \(page 93\)](#) and [Using filters \(page 103\)](#) respectively.

To select specific tables/views:

1. Open the **Select Tables/Views** dialog box.
2. Select a **Schema**.
3. Optionally, select the **Use exact table name** check box. This option is useful if your schema contains numerous tables as it will save you having to scroll through the entire list to find one specific table.
4. If you selected the **Use exact table name** check box, type the exact name of the table you want to replicate in the **Table/View** field.
5. Click **Search**.
The table or tables (If you did not select the **Use exact table name** check box) will be shown in the search results.
6. Select the table by adding it to the list on the right.
7. To add additional tables from the same schema, repeat steps 3-6. To add additional tables from a different schema, repeat steps 2-6.
8. Click **OK** to save your settings.



*If you rename a table in the database, the **Designer** tab will still show the original table name. The **Monitor** tab, on the other hand, will show the new table name.*

Setting load order

You can set the load order for each of the selected tables. This may be useful, for example, if your selected tables list contains tables of different sizes and you want the smaller tables to be loaded before the larger tables. When a group of tables are set with the same load order, Replicate will load the tables according to the table ID.

Load order can be set and modified (see [note](#) below) in the following places:

- The [Select Tables window](#) (opened in Designer view by clicking the **Table Selection** button in the right of the console).
- The **Patterns and Selected Tables** list in the right of the console (in Designer view).



- *Load order cannot be changed while the task is running. If you want to change the load order, first stop the task, then change the load order as desired, and finally reload the target.*
- *Load order cannot be set for "Exclude" patterns.*

To set the load order for a specific table:

1. Select the desired table in the **Selected Tables** list.
2. From the **Load Order** drop-down list, select one of the available priority levels (Lowest Priority, Low Priority, Normal Priority, High Priority, and Highest Priority).

3. This step is only relevant if you are setting load order in the **Select Tables** window. Click **OK** to save your settings and close the **Select Tables** window.

To set the same load order for multiple tables:

1. Select the desired tables in the **Selected Tables** list.
2. From any of the selected items' **Load Order** drop-down list, select one of the available priority levels.
3. This step is only relevant if you are setting load order in the **Select Tables** window. Click **OK** to save your settings and close the **Select Tables** window.

Removing specific tables/views from a replication task

This topic walks you through removing specific tables/views from the replication task.

To remove tables from the Selected Tables list:

1. From the **Selected Tables** list, select a table that you want to remove from the replication task and then click the button with a single left-facing arrowhead (**Remove**).
2. To remove all of the tables/views from the **Selected Tables** or **Selected Tables/Views** list, click the button with two left-facing arrowheads (**Remove All**).
3. Click **OK** to close the **Select Tables** or **Select Tables/Views** dialog box.
4. Click **Save** to make sure that Enterprise Manager saves the table information for this task.

Creating table/view selection patterns

This topic walks you through selecting tables/views using patterns. For example, you can include all tables/views that belong to the HR schema except for one or two tables/views that you exclude. You can also only exclude one or more table/view schemas or tables/views. This replicates the entire endpoint, except for those tables/views that you excluded.

The following example shows a pattern that replicates all tables that are members of the **dbo** schema except for the **dbo.PRODUCT_1%** table.

```
InclUde dbo.%  
ExcLude dbo.PRODUCT_1%
```

You can also use the "_" wildcard character to match a single character. For example, specifying `ExcLude m_d%` will exclude all tables that begin with `m` and end with `d%`, such as `mode1` or `msdb`.



Do not escape wildcard characters as this will instruct Replicate to interpret them as standard characters. As escape character conventions differ across databases, you should consult your database Help for guidance about supported escape characters. Some examples (where an underscore is the wildcard character) are as follows:

- *MySQL and PostgreSQL - \ _*
- *Microsoft SQL Server - [_]*
- *Oracle - For Oracle, use the `escapeCharacter` internal parameter to define a custom escape character.*

When you explicitly select tables/views, all selected tables/views are replicated in full unless you define transformations or filters for the table/view. If you need to make changes to the table/view structures in the target endpoint or if you only want to select specific columns, then you need to perform the procedures described in [Defining transformations for a single table/view \(page 93\)](#) and [Using filters \(page 103\)](#) respectively.



*To view all of the tables/views included when you use a table selection pattern, click the **Full Table List** tab in Designer view. The **Full Table List** lists all of the tables/views included in any table pattern you defined as well as all explicitly selected tables/views. To view only patterns and explicitly selected tables/views, click the **Patterns and Selected Tables** tab in Designer view.*

To create table/view selection patterns:

1. In the Designer view, in the **Select Tables/Views** dialog box, do any of the following:
 - Select a schema from the **Schema** drop-down list. All tables/views that belong to that schema are included in the table/view selection pattern.
 - Type the name or partial name of a table/view in the **Table/View** field. Any string that you enter here is included in the table/view selection pattern.
 - If the table/view that you type here is a member of the schema you selected in the **Schema** drop-down list, then you only have to type the name of the table/view.
 - If you did not select a schema or the table/view belongs to another schema, include the schema with the table name in the following format: `HR.EMPLOYEES`, where HR is the schema.
2. Click **Include** to include all of the tables/views that match the selection criteria.
3. Click **Exclude** to exclude any tables that match the selection criteria.
4. Click **OK** to close the **Select Tables/Views** dialog box.
5. Click **Save** to make sure that Enterprise Manager saves the table/view information for this task.

Excluding specific tables from the replication task

You can easily exclude specific tables from being replicated.

To do this:

1. Open the the **Select Tables/Views** dialog box.
2. Select a **Schema** and then click **Search**.
Any tables in that schema will be shown in the search results.
3. Select the tables by adding them to the list on the right.
4. Click the **Include** button.
Include <schema_name>.% will be added to the **Table Selection Patterns** list.
5. Select the **Use exact table name** check box.
6. Type the name of the table you want to exclude in the **Table/View** field.
7. Click the **Exclude** button.
Exclude <schema_name>.<table_name> will be added to the **Table Selection Patterns** list.
8. To exclude additional tables from the same schema, repeat Steps 6-7. To exclude tables from a different schema, clear the **Use exact table name** check box and then repeat Steps 2-7.
9. Click **OK** to save your settings.



Filters containing wildcard escape characters that excluded/included tables during Full Load will not exclude/include matching tables added during CDC. For example, if there is an exclude pattern `dbo.pc[_]%` and a new table `dbo.pc_table2` is created during CDC, the table will be added to replication task (as opposed to being excluded).

7.5 Editing a replication task

You can make changes to tasks that you previously created. Just open the task and make the changes in the same way that you did when you created the task.

To edit a task:

1. In **Tasks** view, select the task and click **Open**.
The task opens, displaying the source and target endpoints and which tables have been selected for replication.
2. Continue with any of the following procedures:
 - [Adding a source and target endpoint to a task \(page 74\)](#)
 - [Selecting tables and/or views for replication \(page 76\)](#)
 - [Defining transformations for a single table/view \(page 93\)](#)
 - [Using filters \(page 103\)](#)
 - [Task Settings \(page 179\)](#)

7.6 Searching for tasks

In **Tasks** view, you can search for tasks by typing a sequence of letters in the **Search Tasks** box above the tasks. For example, to search for all tasks with names that begin with "Oracle-to", type "or". Only tasks that match the search string are displayed.

7.7 Deleting a replication task

You can delete tasks that you created. To prevent complications, it is recommended not to use the name of a deleted task for a new task you create. Such a task would be created with the same settings as the deleted task.



If you use a Microsoft SQL Server endpoint, a Microsoft SQL Server system administrator must delete the Microsoft SQL Server Replication Publisher definitions for the endpoint that was used in the task from SQL Server.

For more information, see the "Limitations" section in the Microsoft SQL Server chapter in the Qlik Replicate Setup and User Guide.

To delete a task:

1. Stop the task that you want to delete.
2. In **Tasks** view, click **Delete Task**.
The task is deleted.

7.8 Exporting and importing tasks



This functionality is supported with Replicate tasks only.

The ability to export tasks is useful if you need to migrate tasks between different Enterprise Manager machines, which may be necessary if you need to decommission a machine or when moving from a test machine to a production machine, for example. Tasks can be exported with or without endpoints.

Task export use cases

Export Task	When
With Endpoints	<ul style="list-style-type: none"> • The task's endpoints do not exist in the target environment. This way, the task will be created with endpoints when it is imported. • The endpoints already exist in the target environment but with a different configuration that you would like to override. <div style="border: 1px solid #ccc; padding: 10px; margin-top: 10px;">  <p><i>In both of the above cases, after importing the task, you need to edit the endpoints and re-enter the passwords. This will encrypt the passwords using the Master User Key of the target machine.</i></p> </div>
Without Endpoints	The task's endpoints already exist in the target environment with a suitable configuration.

For information on what permissions are required to export and import tasks, see [Roles and permissions \(page 304\)](#).

To export a task:

1. In Tasks view, do one of the following:
 - Select or open the task you want to export and then click the **Export Task** toolbar button.
 - Right-click the task you want to export and select **Export Task** from the context menu.
2. Select **Without Endpoints** or **With Endpoints** accordingly.
Depending on your browser settings, the task JSON file will either be downloaded to your default Downloads folder or you will be prompted to save it to you preferred location.
The file name format is as follows:
AEM_<ReplicateServerName>_<TaskName>_<Date>_<Time>.json

To import a task:

1. If the task is running on the target server, stop the task.
2. In Servers view, do one of the following:
 - Select the target server (i.e. the server to which you want the task to be imported).
 - Right-click the target server and select **Import Task** from the context menu.The **Import Task** window opens.
3. Either select the task JSON file using the **Browse** button or drag the file to the window.
4. Click **Import**.
5. Optionally, when the import completes, start the task.

8 Defining and managing endpoints

Enterprise Manager requires information to connect to the source and target endpoints that you want to use in a task. For a list of endpoints you can work with in Qlik Replicate, see the Qlik Replicate Setup and User Guide.

You use the **Manage Endpoint Connections** window to add endpoints and edit and view the endpoint connection information.



The name cannot exceed 32 characters, contain non-Latin characters, or contain any of the following characters: | \ / : * ? " < >

- [Defining an endpoint \(page 85\)](#)
- [Setting user permissions for a specific endpoint \(page 86\)](#)
- [Editing endpoint configuration information \(page 88\)](#)
- [Viewing endpoint configuration information \(page 89\)](#)
- [Testing an endpoint connection \(page 89\)](#)
- [Duplicating endpoints \(page 90\)](#)
- [Searching for endpoints \(page 90\)](#)
- [Deleting endpoints \(page 90\)](#)

8.1 Defining an endpoint

Before you can begin to design a task, you must add endpoints to the Replicate server. To use an endpoint, you must have access to it somewhere in your system. When you add the endpoint to the Replicate server, you must provide connection information and proper user credentials.

Once you add endpoints to the Replicate server, you can begin to use them to build a replication task. For information on how to add an endpoint to a replication task, see [Adding a source and target endpoint to a task \(page 74\)](#).

To add an endpoint:

1. In the Servers view or on a dedicated task tab, click **Manage Endpoint Connections**.
The **Manage Endpoint Connections** window opens. The server is already selected and cannot be changed.
2. In the **Manage Endpoint Connections** window, click **New Endpoint**.
3. Select the type of endpoint you are using. The information that you must enter depends on which endpoint you select.
For a list of supported endpoints and for more information on setting up a specific endpoint, see the Qlik Replicate Setup and User Guide.

8.2 Setting user permissions for a specific endpoint

This topic explains how to edit user permissions for a specific endpoint, add and remove users or groups, disable or enable inheritance, restore inherited permissions if they were overridden, and view effective permissions for a user.

To do this:

1. In the **Manage Endpoint Connections (<server-display-name>)** dialog, click the **Endpoint Permissions** toolbar button.
The **User permissions for endpoint '<Display-Name>'** dialog opens.
2. See the sections below for the procedures you can perform in the **User permissions for endpoint '<Display-Name>'** dialog.

Adding and removing users

To add a user or group:

1. In the **User permissions for endpoint '<Display-Name>'** dialog, click **Add**.
2. In the **Add User/Group** dialog box, select **User** or **Group**.
3. Enter the name for the new user or group in the following format:
 - NetBIOS_name\user (for example: qa\qa)
 - machine_name/local_user (for example: re2008r2js1\JohnMill)
 - *username* - This format is supported with [SAML authentication only](#). The user/group name can contain any Unicode character up to 255 characters and must be a valid Identity Provider user (Okta or Microsoft Azure).
4. Click **OK** to add the user or group and close the dialog box.
5. Click **OK** to accept the changes, or **Cancel** to undo them.

To remove a user or group:

1. In the **User permissions for endpoint '<Display-Name>'** dialog, select the user or group you want to remove.
2. Click **Remove**.
3. When prompted, click **Yes** to confirm.
4. Click **OK** to accept the changes, or **Cancel** to undo them.

Editing user permissions



Only an Admin can edit user permissions.

To edit a user's permissions:

1. In the **User permissions for endpoint '<Display-Name>'**, adjust the permission slider for a user or group as desired.

8 Defining and managing endpoints



Adjusting the slider stops inheritance from the parent object.

2. Click **OK** to accept the changes or **Cancel** to undo them.

The following table summarizes the roles required for adding and editing the endpoint.

Endpoint operation roles

Operation	Viewer	Operator	Designer	Admin
Add and edit endpoint	No	No	Yes	Yes
View endpoint settings	Partial. Viewers can only see the Name , Description , Role , and Type fields.	Yes	Yes	Yes

Inheritance

By default, inheritance is enabled for all objects (users and groups). This means that permissions are automatically carried over from the parent object. You can turn inheritance on or off for all objects at the current level.

To turn off inheritance:

1. In the **User permissions for endpoint '<Display-Name>'** dialog, click **Disable Inheritance**.



This option disconnects the entire authorization level from the parent level.

2. In the **Disable Inheritance** dialog box, select whether you want to:
 - **Convert inherited permissions on this object into explicit permissions:** This option changes inherited permissions to explicit permissions. Any new users or groups will not inherit permissions from the parent.
 - **Remove all inherited permissions from this object:** This option removes all existing permissions inherited from the parent level. Any new users or groups will not inherit permissions from the parent.
3. Click **Disable**.
If you chose to convert inherited permissions, the check mark in the Inherited column changes into an X. If you chose to remove inherited, all users and groups disappear from the list.
4. Click **OK** to accept the changes or **Cancel** to undo them.

To turn on inheritance:

1. In the **User permissions for endpoint '<Display-Name>'** dialog, click **Enable Inheritance**.



This option enables inheritance for all users and groups on this level.

2. In the **Enable Inheritance** dialog box, select whether you want to:
 - **Inherit all permissions from parent and override any definition manually made at this level:** This option reinstates inherited permissions for all users and groups that are already defined, and new users and groups will inherit their permissions from the parent level.
 - **Inherit all permissions from parent but keep definitions manually made at this level:** This option preserves the permissions already defined for the existing users and groups and adds all permissions from the parent level. New users and groups will inherit permissions from the parent level.
3. Click **Enable**.
4. Click **Save** or **OK** to accept the changes, or **Discard Changes** or **Cancel** to undo them.

To restore inherited permissions for a single user or group if they were overridden:

1. In the **User permissions for endpoint '<Display-Name>'** dialog, select the user or group.
2. Click **Restore Inheritance** .
The check mark returns to the Inherited column to indicate that permissions for this user or group are inherited from the parent.

Viewing effective permissions

Effective permissions are the permissions that are in effect for a user at any particular level.

To view effective permissions for a user:

1. In the **User permissions for endpoint '<Display-Name>'** dialog, do one of the following:
 - Select a user in the list on the left.
 - If a user does not appear in the list but exists in the system and is part of a group, enter the user name in the text field in the **Effective Permissions** pane on the right.
Make sure to use the following format:
 - `NetBIOS_name\user` (for example: `qa\qa`)
 - `machine_name/local_user` (for example: `re2008r2js1\JohnMill`)
2. Click **Get Effective Permissions**.
The effective permissions for the user you entered appear below the button.

For more information on Enterprise Manager's security roles, see [User permissions \(page 298\)](#). For more information on the underlying concepts, see [Granular access control \(page 300\)](#) and [Inheritance and overrides \(page 301\)](#).

8.3 Editing endpoint configuration information

After you add the endpoint to the Replicate server and provide the connection information, you can make changes to some of the information.



You cannot change the following information in the endpoint window:

- The name you provided for the endpoint.
- The endpoint **Type**, for example Oracle or Microsoft SQL Server.
- The endpoint **role**, either **SOURCE** or **TARGET**.

To edit endpoint configuration information:

1. In the **Manage Endpoint Connections** window, select the endpoint you want to edit.
OR
In the **Endpoints** list on the left of the Designer view, double-click the endpoint you want to edit.
Note that this option is only available when editing a specific task.
The **Manage Endpoint Connections** window opens with the selected endpoint settings.
2. Make changes to the information in any of the tabs in the window.
For more information, see the chapter for the specific Qlik Replicate endpoint you are using in the Qlik Replicate Setup and User Guide. For a list of supported endpoints, see the Qlik Replicate Setup and User Guide.

8.4 Viewing endpoint configuration information

After you add the endpoint to the Replicate server and provide the connection information, you can view the information in the **Manage Endpoint Connections** window.

To view endpoint configuration information:

- Select an endpoint from the **Endpoints** list in the left pane; then click the tabs to view the information.

8.5 Testing an endpoint connection

You can try to contact the endpoint to make sure that you are connected to the endpoint you want to work with.

To test the endpoint connection:

1. In the **Manage Endpoint Connections** window, select the endpoint you want to work with.
2. At the bottom of the endpoint's **General** tab, click **Test Connection**.
If the connection is successful, a success message is displayed and a green check mark icon appears next to the **Test Connection** button.
If the connection fails, an error message is displayed at the bottom of the dialog box and the **View Log** button becomes available.
3. If the connection is successful, click **Close**.
If the connection fails, click **View Log** to view the server log entry with information for the connection failure.

8.6 Duplicating endpoints

You can duplicate an endpoint if you need to define a new endpoint with similar settings. Except for the name, all endpoint settings are duplicated to the new endpoint.

To duplicate an endpoint:

1. In the left panel of the **Manage Endpoint Connections** window, click the endpoint you want to duplicate.
2. Click **Duplicate**.
3. On the General tab, edit the name for the endpoint.
4. Make any other necessary changes.
5. Click **Save**; then click **Close**.

8.7 Searching for endpoints

You can search for endpoints by typing a sequence of letters in the **Filter by** box above the endpoints list. For example, to search for all endpoints whose names contain the string "Oracle", type "or". Only endpoints that match the search string are displayed.

8.8 Deleting endpoints

You can delete endpoints that you no longer require. Note that to delete an endpoint that is defined as a source or target in a task, you first need to remove the endpoint from the task.

To delete an endpoint:

- In the left panel of the **Manage Endpoint Connections** window, Select the endpoint and click **Delete**.

9 Customizing tasks

This section describes how to customize a replication task. For example, you can create new tables or columns for the target endpoint or select only some of the data from each column to be replicated. This is done using transformations and filters.



Although the descriptions in this section only refer to tables, the procedures described herein are applicable to views as well. When a transformation is defined for a view, the word "View(s)" appears in the UI instead of the word "Table(s)".

In this section:

- [Table Settings \(page 91\)](#)
- [Defining global rules \(page 119\)](#)
- [Using the Expression Builder \(page 149\)](#)
- [Task Settings \(page 179\)](#)

9.1 Table Settings

In the **<Table_Name> - Table Settings** window, you can define how the data for each individual table/view is replicated to the target.



Some of the table settings are not available in a Log Stream Staging setup.

For information on the availability of table settings in a Log Stream Staging setup, refer to the [Qlik Replicate Setup and User Guide](#).

To open the Table Settings window:

1. Open the task you are working with.
For information on opening a task, see [Editing a replication task \(page 82\)](#).
2. In Designer view, select the desired table from one of the following tabs on the right of the console:
 - The **Patterns and Selected Tables** tab - if the desired table was explicitly selected.
 - The **Full Table List** tab - if the desired table was selected using a table inclusion pattern.
For information on how to define table selection patterns, see [Creating table/view selection patterns \(page 80\)](#).
3. Click the **Table Settings** button above the table list.
The **<Table_Name> - Table Settings** window opens.
4. In the **Table Settings** window, perform any of the following tasks:
 - [Performing General tasks for a single table/view \(page 92\)](#)
 - [Defining transformations for a single table/view \(page 93\)](#)
 - [Using filters \(page 103\)](#)

- [Parallel Load \(page 110\)](#)
 - [Handling LOB columns \(page 114\)](#)
 - [Message format \(page 118\)](#)
 - [Full Load \(page 119\)](#)
5. Click **OK** to close the **Table Settings** window.
 6. Click **Save** in the main toolbar to preserve the table and column information for this task.

To restore the default table values:

- Click **Restore Table Defaults** at the bottom left of the **Table Settings** window. This option is available in all tabs.
Any changes you made will be discarded and the table's default settings will be restored.



*The names of modified tables will be followed by the word (**changed**), enabling you to easily identify which tables have been modified.*

Performing General tasks for a single table/view



Although the descriptions in this section only refer to tables, the procedures describe herein are applicable to views as well. When a task is being performed for a view, the word "View(s)" will appear in the UI instead of the word "Table(s)"

The **General** tab in the Table Settings window displays basic information about the selected table and allows you to define new names for the table/schema on the target as well as override the default tablespace for the table and its index (Oracle target only).

To edit the general table settings:

1. Open the [Table Settings \(page 91\)](#) window.
2. Click the **General** tab on the left side of the window, as shown below.

DEMO.EMPLOYEE - Table Settings □ ×

General

Transform

Filter

Parallel Load

LOB Column Handling

Source Table Details

Table Schema	DEMO
Table Name	EMPLOYEE
Record identifier	Non clustered primary key
Identifier key/index name	_SYS_TREE_RS_#8726962_#0_#P0
Table tablespace	
Index tablespace	
Approximate record size (Byte)	79
Estimated record count	15
Estimated table size (MB)	0
Load Order	Normal Priority

Map to target table

Table Schema:

Table Name:

Table tablespace:

Index tablespace:

↻ Restore Table Defaults OK Cancel

In the **Map to target table** section, the following options are available:

- **Table Schema:** Specify the schema in which you want the table to be created on the target.
- **Table Name:** Specify a new name for the table on the target.
- **Table tablespace:** This option is only available when the task is defined with an Oracle target endpoint.
Specify the name of the tablespace in which you want the table to be created on the target. By default (i.e. when this field is empty), the table will either be created in the source table tablespace on the target (when replicating from an Oracle source) or in the default tablespace (when replicating from any other source).
- **Index tablespace:** This option is only available when the task is defined with an Oracle target endpoint.
Specify the name of the tablespace in which you want the table's index to be created on the target. By default (i.e. when this field is empty), the index will either be created in the source table tablespace on the target (when replicating from an Oracle source) or in the default tablespace (when replicating from any other source).

Defining transformations for a single table/view



Although the descriptions in this section only refer to tables, the procedures describe herein are applicable to views as well. When a transformation is defined for a view, the word "View(s)" will appear in the UI instead of the word "Table(s)".

This section describes how to define data transformations. Data transformations are performed when the task is run. They are optional. If you do not define any transformations, the data is replicated "as is" from the source to the target.

Enterprise Manager lets you make the following changes to the tables and columns:

- Rename any column for the target table
- Delete a target column
- Change the data type and/or the length of any target column
- Add additional target columns
- Designate which target columns (i.e. segments) will comprise the Unique Index
- Recalculate the data

Limitations

Transformations are subject to the following limitations:

- Transformations are not supported for columns with right-to-left languages.
- By default, transformations cannot be performed on columns that contain special characters (e.g. #, \, /, -) in their name.
However, such functionality is provided via the **Support special characters in column names used in expressions** in the [task settings](#). If you intend to enable option, make sure to carefully review the requirements first.
- Transformations cannot be performed on columns that contain special characters (e.g. #, \, /, -) in their name.
- The only supported transformation for LOB/CLOB data types is to drop the column on the target.
- Using a transformation to rename a column and then add a new column with the same name is not supported.

You can use the method described here for transformations that are specific to a single table or a few tables in your task. To make a similar change over multiple tables, see [Starting the Global Transformation Rules wizard \(page 120\)](#).

For an explanation of how to configure transformations, see [Using the Transform tab \(page 95\)](#).

To define a data transformation for a single table:

1. Select the table you want to transform and open the [Table Settings \(page 91\)](#) window.
2. Click **Transform** on the left side of the window.

The following figure shows the information in the **Transform** tab of the **Table Settings** window.

HR.DEPARTMENTS Table Settings
✕

General

Transform

Filter

Source table: HR.DEPARTMENTS By default include all columns

Input

DEPARTMENT_ID

DEPARTMENT_NAME

MANAGER_ID

LOCATION_ID

>

<

>>

<<

Output Add Column

Key	Name	Type	Expression
🔑	DEPARTMENT_ID	INT2	✎
	DEPARTMENT_NAME	STRING(30)	✎
	MANAGER_ID	INT4	✎
	LOCATION_ID	INT2	✎

↻ Restore Table Defaults
OK
Cancel

Using the Transform tab

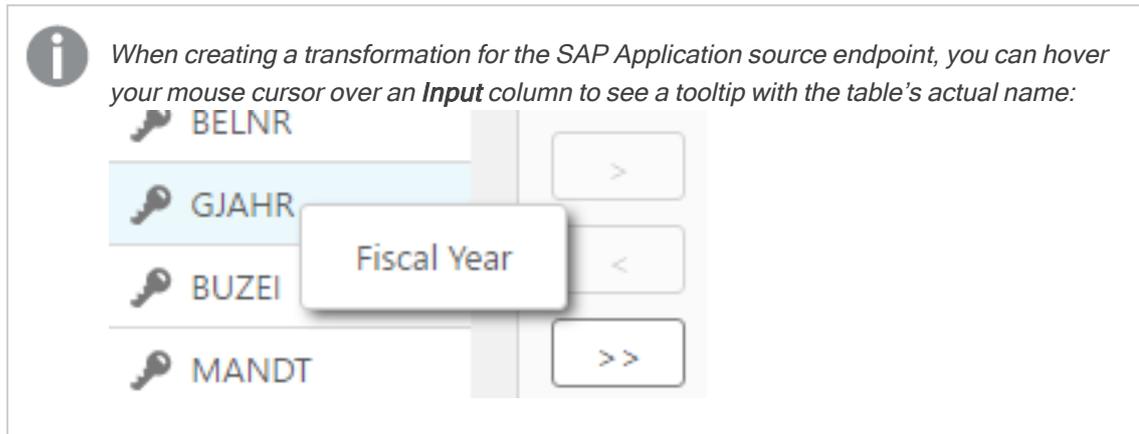
In the **Transform** tab, you can define transformations using Replicate's built-in functionality.



Customers that requires functionality not provided by Replicate's built-in transformations can write their own transformations, and then access them from the [Replicate Expression Builder](#). For an explanation of how to create user-defined transformations (requires basic programming skills), see [User-defined transformations \(page 177\)](#).

The **Transform** tab in the **Table Settings** window consists of the following elements:

- **Input:** This lists the columns on which you can perform transformations.



- **Output:** This table shows the defined output for the columns in the table where you are performing the transformation(s). See Transformation Options below for information on how to change the default output.

Limitations and considerations

- Dropping a column, saving your changes, and then adding a column with the same name and defining an expression corresponding to the dropped column's data, is not supported. If you mistakenly drop a column, simply add the column back again without an expression.
- If you stop a task and define a metadata transformation for one of the tables (such as dropping a column), make sure the **DROP and CREATE table** option is selected (the default) in the Task Settings' **Full Load Settings** tab before resuming the task.
- In homogeneous replication tasks (such as Oracle to Oracle), modifying a single table column (by changing the column data type or length for example), will break the homogeneity for the entire table.
- Transformation of numeric data types must fall between the range -9223372036854775808 to +9223372036854775807.

Transformation options

The following table describes the transformation options available in the **Transform** tab.

Transform actions

To	Do This
Rename a column	<p>Select the Name column for the table column you want to change. Type in a new name.</p> <p>The top right corner turns blue when the name is changed. To view the original name, hover the mouse pointer over the field and the original name is displayed.</p>

To	Do This
Use the Primary Key constraints that exist in the target table instead of the source table's Primary Key constraints	<ol style="list-style-type: none"><li data-bbox="555 293 1326 398">1. Click the Set Key Column button. The Primary Key Columns dialog will open showing the source table's primary key columns.<li data-bbox="555 412 927 443">2. Select Use target definition. <div data-bbox="600 456 1390 987" style="border: 1px solid #ccc; padding: 10px;"><p data-bbox="619 479 1358 636"> <i>If the target table already exists and you want to retain its Primary Key constraints, then you must set one of the following options in the task settings' Full Load Settings tab:</i></p><ul data-bbox="751 651 1337 763" style="list-style-type: none"><li data-bbox="751 651 1337 719">• <i>If target table already exists, TRUNCATE before loading</i><li data-bbox="751 730 1238 763">• <i>If target table already exists, Do nothing</i><p data-bbox="700 775 1358 954"><i>If you do not select these options, the table will initially be created with the source table's Primary Key constraints. After the target table is created, any changes to its Primary Key constraints will be used instead of the source table's Primary Key constraints.</i></p></div><li data-bbox="555 1010 1382 1077">3. Click OK to save your changes and close the Primary Key Columns dialog.

To	Do This
Define different Primary Key columns on the target and optionally change their order	<p>Replicate relies on primary key columns (or indexes) defined in the target tables to be able to correctly apply changes to them.</p> <p>In some cases - for example, if the target table consolidates data from multiple sources - you might need to define different Primary Key columns on the target table and arrange them in a specific order to improve performance.</p> <div data-bbox="520 566 1390 815" style="border: 1px solid #ccc; padding: 10px; margin: 10px 0;"> <p> <i>When upgrading, to preserve the behavior of existing tasks (as opposed to new tasks), this improvement is turned off by default. To turn it on for existing tasks, after upgrading, open the task settings and either delete the <code>use_manipulation_pk_for_apply</code> parameter from the More Options tab or set the value to Off.</i></p> </div> <p>To add Primary Key constraints to the target table and arrange them in a specific order:</p> <ol style="list-style-type: none"> 1. Click the Set Key Column button. The Primary Key Columns dialog opens showing the source table's primary key columns. 2. Select Use transformation definition. 3. To add a Primary Key constraint on a field, click the cell in the Key column. A key icon will be added to the Key column with a number indicating the order of the Primary Key constraint. Repeat to add Primary Key constraints to other fields. 4. To remove a Primary Key constraint from a specific field, click the key icon in the Key column. 5. To change the order of a Primary Key column, select a Primary Key column and use the Up/Down arrows. 6. Click OK to save your changes and close the Primary Key Columns dialog. A number will now appear next to each of the Primary Key columns, indicating how they will be ordered in the target table. <div data-bbox="520 1621 1390 2063" style="border: 1px solid #ccc; padding: 10px; margin: 10px 0;"> <p> <i>When any of the following options are selected in the task settings' Full Load Settings tab, the table will be created with the defined keys:</i></p> <ul style="list-style-type: none"> • <i>If target table already exists, DROP and CREATE table</i> • <i>If target table already exists, ARCHIVE and CREATE table</i> <p><i>When any of the following options are selected in the task settings' Full Load Settings tab, Replicate will use the defined keys to determine which target columns are Primary Keys (as opposed to setting them in the actual target table):</i></p> <ul style="list-style-type: none"> • <i>If target table already exists, TRUNCATE before loading</i> • <i>If target table already exists, Do nothing</i> </div>

To	Do This
Change the data type for a column	<p>Select the Type column for the table column you want to change and select a new data type from the drop-down list. Make sure that the data type you select is compatible with the data in that column.</p> <p>For a description of Qlik Replicate data types, information about data-type mapping from the native endpoint to Qlik Replicate, and for a list of endpoints supported by Qlik Replicate, see the Qlik Replicate Setup and User Guide.</p>
Change the data subtype for a column	<p>This option is available for the CLOB, NCLOB, STRING, and WSTRING data types only.</p> <p>Select the Subtype column for the table column whose data type you want to change, and then select either JSON or XML from the drop-down list. Make sure that the data in the column is compatible with the selected subtype. The default is Regular, which means that data type in the Type column will be used with no subtype.</p> <p>For a description of Qlik Replicate data types, information about data-type mapping from the source endpoint to Qlik Replicate, and for a list of endpoints supported by Qlik Replicate, see the Qlik Replicate online help.</p>
Add a new column	<p>Click Add Column to add a new column. When you add a column, the Name is blank and the Type is listed as <code>string(50)</code>.</p> <p>Type a name for the new column in the Name column. If needed (according to the column data), click in the Type column and select a data type from the list.</p>
Add an existing column	<p>From the Input pane, select one or more columns and click the right facing arrow button.</p> <p>To add all of the columns, click the right-facing double arrow.</p> <div data-bbox="520 1429 1390 1865" style="border: 1px solid #ccc; padding: 10px; margin-top: 10px;"> <p> • <i>By default all tables columns are included in the Output list. To include only some of the columns clear the By default include all columns check box at the top of the Transform tab. This removes all of the columns from the list. You can then add back any existing column as required.</i></p> <p>• <i>If a column is explicitly added at the table level but then dropped at the global level (using a global transformation rule), the column will still be created on the target, but without any data.</i></p> </div>

To	Do This
Delete a column	<p>From the Output list, select the row with the column you want to delete and click the left-facing arrow button.</p> <p>To remove all columns, click the left-facing double arrow. Note that all the columns except for columns defined as a primary key/unique index are deleted.</p>
Recalculate the data for a column in the target endpoint	<p>Click in the Expression column in the row with the table column you want to change the data for. Enter an expression using SQLite syntax.</p> <p>See Creating an expression for transformations (page 102) and Using SQLite syntax with transformations (page 102) for information on creating expressions.</p> <p>Once you add a calculated expression, you can test the expression. See Using the Expression Builder (page 149).</p>
<p>Change the data type for a specific input column</p> <div data-bbox="204 949 488 1312" style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> <p> <i>Supported with the IBM DB2 for iSeries and IBM DB2 for z/OS source endpoints only.</i></p> </div>	<p>This is required if a source column is defined as character type but the data stored in that column is binary or vice versa.</p> <div data-bbox="520 949 1388 1122" style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> <p> <i>When the source column type is STRING, WSTRING, CLOB, or NCLOB, you must also select a Character Set, otherwise an error will be shown and the OK button will be disabled.</i></p> </div> <p>In the Input table, click the relevant cell in the Type column and then select either STRING or BYTES from the drop-down list as required.</p> <div data-bbox="520 1249 1388 1422" style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> <p> <i>If you change a column's Type in the Input table, you also need to set the same Type for the corresponding column in the Output table.</i></p> </div> <p>Note that if you select STRING, you can also change the character set, as explained below.</p> <div data-bbox="520 1550 1388 1682" style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;"> <p> <i>Modified cells will display a triangle in the top right corner. To see the original value, click the triangle.</i></p> </div>

To	Do This
<p>Change the Character Set for a specific input column</p> <div data-bbox="204 427 488 790" style="border: 1px solid gray; padding: 5px; margin-top: 10px;"> <p> <i>Supported with the IBM DB2 for iSeries and IBM DB2 for z/OS source endpoints only.</i></p> </div>	<p>This is required if a source character column is wrongly encoded. For example, if a source character column is described as encoded in CCSID X, but the data stored in that column is actually encoded in CCSID Y.</p> <p>You can also set a custom character set as described in Setting a custom character set (page 101) below.</p> <p>In the Input table:</p> <ol style="list-style-type: none"> 1. Click the relevant cell in the Type column and select STRING from the drop-down list. 2. Click the relevant cell in the Character Set column and then select the appropriate character set from the drop-down list. <div data-bbox="603 757 1386 972" style="border: 1px solid gray; padding: 5px; margin-top: 10px;"> <p> • <i>Only character sets compatible with the selected Type will be available for selection.</i></p> <p>• <i>Modified cells will display a triangle in the top right corner. To see the original value, click the triangle.</i></p> </div>

Setting a custom character set

The following procedure is supported with the IBM DB2 for iSeries and IBM DB2 for z/OS source endpoints only.

Perform the steps below if the source table is defined with an incorrect CCSID and the correct definition is actually in a UCM file.

1. Create a mapping data file with the file extension **.ucm**.

 *If you edit an existing UCM file, you must also change the values of the `<code_set_name>` and `<icu:alias>` properties. If the file does not contain an `<icu:alias>` property, then you only need to change the value of the `<code_set_name>` property.*

2. Create a CNV file for the UCM file by running the following command:

```
<product_dir>\bin\makeconv.exe -v <file_name>.ucm
```

Example:

```
"c:\Program Files\Attunity\Replicate\bin\makeconv.exe" -v 1047_EX.ucm
```

This will create a CNV file with the same name as the UCM file (for example, `1047_EX.cnv`).

3. Create a new subfolder named **icudt<XX>I** under `<product_dir>\bin` where the **XX** is the same as the number in the **icudt<XX>.dll** file name.
For example, If the DLL file name is **icudt69.dll**, create a new subfolder named **icudt69I**.
4. Copy the CNV file to the subfolder you just created (`<product_dir>\bin\icudt69I` in the example above).



When using the Replicate File Channel, the file should be copied to the same location on both Replicate servers.

5. Restart the Qlik Replicate UI Server service.
5. Select the custom character set from the **Character Set** drop-down list; it will appear as the CNV file name followed by the word "Custom" e.g. **1047_EX.cnv (Custom)**.

Using a column's before-image data in a transformation

You can use a column's before-image data in a transformation. This is useful if you need to store the before-image data on the target.

To do this, simply specify the source column name in **Output** table's **Expression** column, in the following format:

```
$BI__MyColumn
```

Where `$BI__` is a mandatory prefix (that instructs Replicate to capture the before-image data) and `MyColumn` is the source column name.

Although you can store the before-image data in an existing target column, it is recommended to create a new target column (using the **Add Column** button) in which to store the before-image data.

Creating an expression for transformations

Use an expression to define the contents of a new or re-calculated column.

To create an expression:

1. In the **Transform** tab, select the row with the column for which you want to create an expression.
or
Click **Add Column** to add a new column.
2. Click the **fx** button in the **Expression** column.
The **Expression Builder** opens.
3. Build an expression as described in [Using the Expression Builder \(page 149\)](#).

Using SQLite syntax with transformations

The following table lists the SQLite operators that are supported with transformations.

SQLITE syntax operators

Operator	Description
	Concatenate strings. FIRST_NAME LAST_NAME PHONE_NUMBER <Office Only> (adds the string Office Only to the telephone number).

Operator	Description
+	Adds two values together. DEPARTMENT_ID+100 (adds 100 to each ID number). Any column used in an expression with this operator must be a numeric data type.
-	Subtracts a value from another value. MANAGER_ID-100 (subtracts 100 from each ID number). Any column used in an expression with this operator must be a numeric data type.
%	Uses the remainder of a division expression as the value. %SALARY/7 (Divides the value of the Salary column by 7 and uses any remainder from the expression as the column value).
/	Divides one value into another. SALARY/.16 (Divides the value of the Salary column by .16. <div style="border: 1px solid gray; padding: 10px; margin-top: 10px;">  <i>If the two values in the division expression are integers (two NUMERIC columns with no digits after the decimal) and the result is a fractional value, the result returned will be 0.</i> </div>
*	SALARY*.16 (Multiplies the value of the Salary column by .16. This could be used to calculate taxes that are subtracted from a salary).

For more information about SQLite syntax, see the SQLite documentation.

Using filters

Filters let you include or exclude records from a replication task based on the value(s) of the source table columns, thereby allowing you to replicate only the specific data that you need.

In this section:

- [Filter limitations \(page 103\)](#)
- [Opening the Filter tab \(page 104\)](#)
- [Creating a filter condition for a specified column \(page 105\)](#)
- [Creating a record selection condition for one or more columns \(page 105\)](#)
- [Adding or removing filter ranges \(page 107\)](#)
- [Using SQLite syntax with filtering \(page 108\)](#)

Filter limitations

When creating a filter, the following limitations apply:

- Filters are not supported for calculating columns of Right-to-Left languages.
- Filters can only be applied to immutable columns.

- **Filters on mutable columns:**
When a filter is created to exclude/include specific rows in a column, the specified rows will always be excluded/included, even if the rows that were originally excluded/included are later changed. For example, if you chose to exclude/include rows "1-10" in a column named "Age" and those rows were later changed to "11-20", the rows will continue to be excluded/included, even though the data is no longer the same.
Additionally, if a row outside the filter scope was changed (i.e. updated or updated and then deleted) so that it should now be excluded/included (as defined by the filter), it will not be replicated to the target. So, for example if you created a filter to exclude/include rows less than 5 and then changed row 6 to -6, it will not be replicated (even though it is included in the filter's criteria range).
- Filter cannot be applied to LOB columns.
- When specifying numeric data as a filtering condition, the data preceding the decimal point cannot exceed int64.

Opening the Filter tab

The Filter Table tab contains the following information:

- **Data Columns** list: This list contains a list of the columns for the table where you filtering data. You can use these to select the columns to use in the filtering operations.
This list has the following tabs:
 - **Source:** This tab lists the original source columns in the table.
 - **Header:** This tab lists the available header columns. You can create filters using these columns and include them in expressions. For information on these header columns, see [Headers \(page 173\)](#).
 - **Calculated:** This tab lists the columns added to the table. You add columns through transformations. For more information, see [Defining transformations for a single table/view \(page 93\)](#).
- **Filter Conditions** table: This table has the following columns:
 - **Name:** The name of the column where you are filtering the data.
 - **Type:** The data type for the column.
 - **Include/Exclude:** Indicate whether to include or exclude the filtered data for this column.
 - **Ranges:** Click the button on the right of the **Ranges** field to open the Range Builder. For information on creating a value or ranges with the Range Builder, see [Adding or removing filter ranges \(page 107\)](#).
For more information on typing in the filter ranges manually, see [Using SQLite syntax with filtering \(page 108\)](#).
- **Record Selection Condition:** Enter a complex condition that can include multiple columns. The condition must evaluate to TRUE to be accepted. You can create a condition using SQLite operators or by [Using the Expression Builder \(page 149\)](#). For information on using the SQLite operators, see [Creating a record selection condition for one or more columns \(page 105\)](#).

The following figure is an example of the information in the **Filter** tab of the **Table Settings** window.

Table Settings: Filter

HR.DEPARTMENTS Table Settings

General
Transform
Filter

Data Columns

Source	Header	Calculated
DEPARTMENT_ID		
DEPARTMENT_NAME		
MANAGER_ID		
LOCATION_ID		

>
<
>

Filter Conditions

Name	Type	Include/Exclude	Ranges
DEPARTMENT_NAME	STRING(30)	Include	

Record Selection Condition

Expression Builder

Restore Table Defaults OK Cancel

To open the Filter tab:

1. Select the table you want to filter and then open the [Table Settings \(page 91\)](#) window.
2. Click the **Filter** tab on the left side of the window.

Creating a filter condition for a specified column

You can create a simple condition for a single column in the table you are working with. You can include any combination of ranges or specific values in the filter and determine whether to include or exclude the defined data.

To create a filter condition:

1. Select a column from the data columns list and then click the right-facing arrow next to the **Filter Conditions** table.
To remove the column, click on it in the **Filter Conditions** table and then click the left-facing arrow. Any data entered for this column in the **Include/Exclude** or Values columns is also deleted.
2. Click in the **Include/Exclude** column to select whether to include or exclude the data that meets this condition.
3. Click the Edit Ranges button in the **Ranges** column.
4. The **<Name> <Include/Exclude> Ranges** window opens. Continue from [Adding or removing filter ranges \(page 107\)](#).

Creating a record selection condition for one or more columns

You can create a record selection condition manually and/or by using the Expression Editor.

When entering a string, you can use the following special characters:

- `%`: Matches any string of zero or more characters. For example, `Mc%` searches for every name that begins with **Mc** or `%bob%` includes every name that contains **bob**.
- `_`: Matches a single character (as a wildcard). For example: `'sm_th'` includes names that begin with **Sm** and end with **th**, such as **Smith** or **Smyth**. To search for an underscore character, use `[_]`.
- `[. .]`: Includes a range or set of characters. For example, `[CK]ars[eo]` includes names **Karsen**, **Karsen**, **Carson**, and **Karson** or `[M-Z]inger` includes all words that end in **inger** with the first letter between **M** and **Z**, such as Ringer, Singer, or Zinger.

For more information, see documentation on how to use Transact-SQL.

For information on what SQLite operators can be used to create Record Selection Condition filters, see [Using SQLite syntax with filtering \(page 108\)](#).

To create a record selection condition:

1. From the **Data Columns** list, select a source column, header column or calculated column and then click the arrow to the left of the **Record Selection Condition** pane.
2. Use SQLite operators, such as `<` or `=` to create the condition. Use any amount of strings or columns as you need to create a condition.

For example `$EMPLOYEE_ID < 100 AND $SALARY > 100,000`

In this case only rows that satisfy both of these conditions are replicated in the replication task.

The following example provides an example using SQL search pattern strings. Only rows that satisfy this condition are replicated.

```
$EMPLOYEE_NAME IS 'sm_th'
```

To create a record selection condition using the Expression Builder:

- Click **Open Expression Builder**. This button is located directly under the record selection condition box. Follow the directions for creating an expression in the section [Using the Expression Builder \(page 149\)](#).

Applying updates to specific columns only

You can define an expression that instructs Replicate only to apply UPDATES when a user-defined condition has been met, for example, only when specific columns have changed. This is useful in situations when there are many updates in the source that the user has deemed not relevant for the target, as "irrelevant" updates will be ignored.

Limitations

- Does not support columns that do not have Before-Image data (e.g. LOB columns)
- Does not support the following sources (i.e. sources that do not support Before-Image records):
 - ODBC with CDC
 - Teradata
- Subject to the existing expression builder and filter limitations
- Content-based filtering may result in loss of data or data corruption. For example, if the Primary Key value changes (an UPDATE operation), the expression may ignore the UPDATE if the columns that were specified in the expression did not change. The result in this case would be that a "phantom"

row with the old row contents will remain, even if a change was later applied to the columns specified in the expression.

How to

Assume that you have a source table named **table1** with columns **c1-c10** but you are only interested in updating columns **c7**, **c8** and **c9** on the target.

To do this, you would need to:

1. [Open the Table Settings](#) for **table1** and select the **Filter** tab.
2. Click the **Expression Builder** button at the bottom right of the tab. The Expression Builder opens.
3. Optionally, select the **Headers** tab.



*Although selecting the **Headers** tab is optional, selecting it will enable you to add `$AR_H_OPERATION` to your expression (as required in Step 4 below) simply by double-clicking the column on the left of the tab.*

4. Enter the following expression in the **Build Expression** pane and then click **OK**:
`($AR_H_OPERATION != 'UPDATE') OR ((($AR_H_OPERATION = 'UPDATE') AND ((ifnull($BI__c7,0) != ifnull($c7,0)) OR (ifnull($BI__c8,0) != ifnull($c8,0)) OR (ifnull($BI__c9,0) != ifnull($c9,0))))`

The above expression means that changes will be applied to **c7**, **c8** and **c9** only if one of the following is true:

- The operation is not an UPDATE.
- The value of **c7**, **c8** or **c9** has changed as the result of an UPDATE operation.



When used in an expression, Before-Image columns must be prefixed with `$BI__`. For operations other than UPDATE, the value of the specified columns will be NULL.

Adding or removing filter ranges

You can add one or more values to the Ranges column using the Range Builder. Values that match any of the ranges in the list are included in the replication.

You can also delete a filter range using the Range Builder.



Filter ranges that you enter manually are also displayed in the Filter Builder. You can use the Filter Builder to delete them.

To use the Range Builder:

1. In the **Filter** tab of the [Table Settings \(page 91\)](#) window, select a column to filter. For more information, see [Using filters \(page 103\)](#).
2. Click the button to the right of the **Ranges** column.
The **Ranges Builder** opens.
3. Click **Add Range**. Select any of the following from the drop-down list displayed.
 - **Equal to**: Select **Equal to** to enter a single value. The following is displayed in the range list.
Equal to = [N]
Click the **[N]** and type a value in the field that is displayed.
When the value in the selected column equals the value you enter, the result is included or excluded in the replication task depending on the option selected in the **Include/Exclude** column.
 - **Between**: Click **Between** to enter a range of values. The following is displayed in the range list.
Between [N] - [N]
Click each **[N]** and type a value in the fields that are displayed.
When the column contains the values between the two values entered, the result is included or excluded in the replication task depending on the option selected in the **Include/Exclude** column.
 - **Less than or equal to**: Select **Less than or equal to** and enter a maximum value. The following is displayed in the range list.
Less than or Equal to =< [N]
Click the **[N]** and type a value in the field that is displayed.
When the value in the selected column is equal to or less than the value you enter, the result is included or excluded in the replication task depending on the option selected in the **Include/Exclude** column.
 - **Greater than or equal to**: Select **Greater than or equal to** and enter a minimum value. The following is displayed in the range list.
Greater than or Equal to => [N]
Click the **[N]** and type a value in the field that is displayed.
When the value in the selected column is equal to or more than the value you enter, the result is included or excluded in the replication task depending on the option selected in the **Include/Exclude** column.

To delete a filter range from the Range Builder:

1. In the **Filter** tab of the [Table Settings \(page 91\)](#) window, select the column with the filter condition you want to delete.
2. Click the button to the right of the **Ranges** column. The Ranges Builder opens.
3. Click the **X** next to the range you want to delete. The deleted range is removed from the list.

Using SQLite syntax with filtering

Qlik Replicate supports the following SQLite operators when creating Record Selection Condition filters.



You must put the (\$) in front of each input as shown below.

SQLITE syntax operators

Operator	Description
<	Is less than. <code>\$SALARY<100000</code>
<=	Is less than or equal to <code>\$SALARY<=100000</code>
>	Is greater than <code>\$SALARY>100000</code>
>=	Is more than or equal to <code>\$SALARY>=100000</code>
=	Is equal to <code>\$SALARY=100000</code>
!= or <>	Is not equal to <code>\$SALARY!=100000</code>
IS	Is the same as <code>\$HIRE_DATE IS 2014-09-29</code> IS functions the same as = unless one or both of the operands are NULL. In this case, if both operands are NULL, then the IS operator evaluates to 1 (true). If one operand is NULL and the other is not, then the IS operator evaluates to 0 (false).
IS NOT	Is not the same as <code>\$HIRE_DATE IS NOT 2014-09-29</code> IS NOT functions the same as != unless one or both of the operands are NULL. In this case, if both operands are NULL, the IS NOT operator evaluates to 0 (false). If one operand is NULL and the other is not, then the IS NOT operator evaluates to 1 (true).
AND	Both operands are true. <code>\$MANAGER_ID AND EMPLOYEE ID >100</code>
OR	Either operand is true. <code>\$MANAGER_ID OR EMPLOYEE ID >100</code>

For more information on how to use the SQLite syntax, see the SQLite documentation.

Parallel Load

In Full Load replication mode, you can accelerate the replication of large tables by splitting the table into segments and loading the segments in parallel. Tables can be segmented by data ranges, by partitions, or by sub-partitions.

Supported endpoints

The task must be defined with a combination of the following source and target endpoints:

Supported source endpoints:

- Amazon RDS for Microsoft SQL Server
- IBM DB2 for LUW
- IBM DB2 for z/OS



Table segmentation by partitions or sub-partitions is not supported with the IBM DB2 for z/OS source endpoint.

- Microsoft SQL Server
- MySQL
- Oracle
- PostgreSQL



Table segmentation by partitions or sub-partitions is not supported with the PostgreSQL source endpoint.

- SAP Sybase ASE
- SAP Application
- SAP Application (DB)



Tables are by default client dependent with the SAP Application (DB) source endpoint. The MANDT column is automatically taken directly from the endpoint.

- SAP HANA
- Snowflake
- Teradata

Supported target endpoints:

- Amazon EMR
- Amazon MSK
- Amazon Redshift
- Amazon S3

- Cloudera Data Platform (CDP) Private Cloud
- Confluent Cloud
- Databricks (Cloud Storage)
- File
- Google Cloud BigQuery
- Google Cloud SQL for MySQL
- Google Cloud SQL for PostgreSQL
- Google Cloud Storage
- Google Dataproc
- Hadoop (Hortonworks and Cloudera)
- Hortonworks Data Platform (HDP)
- Kafka
- Microsoft Azure ADLS
- Microsoft Azure Data Warehouse
- Microsoft Azure Event Hubs
- Microsoft Azure HDInsight
- Microsoft Azure SQL Database
- Microsoft Fabric Data Warehouse
- Microsoft SQL Server
- MySQL
- Oracle
- PostgreSQL
- Snowflake on Google
- Snowflake on AWS
- Snowflake on Azure
- Sybase ASE



*To prevent deadlocks when performing a full parallel load of partitioned tables into Sybase ASE, it is strongly recommended to enable the **Create primary key or unique index after full load completes** option in the Full Load settings.*

Setting up Parallel Load

To define segment boundaries by data range:

1. In the **Parallel Load** tab's **Select Parallel Load Method** section, select **Use Data Ranges**.
2. In the **Select Details** section, click **Select Segment Columns**.
The **Columns** window opens
3. For all endpoints, the Unique Index column is automatically selected. Select which additional columns whose data you wish to use to delineate the ranges and then click **OK**.

i

- *Selecting indexed columns will significantly improve performance*
- *You can select up to ten columns (multi-selection is supported)*
- *Records with null values will not be replicated*
- *The following data types cannot be used to define segments by ranges: DOUBLE, FLOAT, and LOB (BLOB, CLOB, NCLOB)*

4. In the **Define Segment Boundaries** section:

- Click **Add Segment** to add a segment.
The columns that you selected will appear as table headings.
- Enter the upper data range for the segment in the selected columns.

i *Values in DATE columns must be entered in the format supported by the source. For example, for an Oracle source, the correct format would be:*

- *ALTER SESSION SET NLS_DATE_FORMAT: 'YYYY-MM-DD HH24:MI:SS' (specifying YYYY-MM-DD only is also valid)*
- *ALTER SESSION SET NLS_TIMESTAMP_FORMAT: 'YYYY-MM-DD HH24:MI:SS.FF9'*
- *ALTER SESSION SET NLS_TIMESTAMP_TZ_FORMAT: 'YYYY-MM-DD HH24:MI:SS.FF9 TZH:TZM'*

- Add additional segments as required.
- Click **Validate** to validate that the specified data corresponds to the source column data type and that all of the defined segments contain values.
- To delete a segment, select the desired segment and then click **Delete**.

5. Click **OK** to save your settings.

i *When **Use Data Ranges** is selected, all of the table data will be replicated, even if data ranges are not defined for all of the columns.*

Usage example

Let's assume that the following segments are defined in the **Define Segment Boundaries** table:

Example table data

Column_1	Column_2	Column_3
10	30	105
20	20	120
100	12	99

In this case, the following "WHERE" clauses will be created for each load segment:

- **Segment 1:** $((COL1 < 10) \text{ OR } ((COL1 = 10) \text{ AND } (COL2 < 30)) \text{ OR } ((COL1 = 10) \text{ AND } (COL2 = 30) \text{ AND } (COL3 < 105)))$
- **Segment 2:** $\text{NOT } ((COL1 < 10) \text{ OR } ((COL1 = 10) \text{ AND } (COL2 < 30)) \text{ OR } ((COL1 = 10) \text{ AND } (COL2 = 30) \text{ AND } (COL3 < 105))) \text{ AND } ((COL1 < 20) \text{ OR } ((COL1 = 20) \text{ AND } (COL2 < 20)) \text{ OR } ((COL1 = 20) \text{ AND } (COL2 = 20) \text{ AND } (COL3 < 120)))$
- **Segment 3:** $\text{NOT } ((COL1 < 20) \text{ OR } ((COL1 = 20) \text{ AND } (COL2 < 20)) \text{ OR } ((COL1 = 30) \text{ AND } (COL2 = 20) \text{ AND } (COL3 < 120))) \text{ AND } ((COL1 < 100) \text{ OR } ((COL1 = 100) \text{ AND } (COL2 < 12)) \text{ OR } ((COL1 = 100) \text{ AND } (COL2 = 12) \text{ AND } (COL3 < 99)))$
- **Segment 4:** $\text{NOT } ((COL1 < 100) \text{ OR } ((COL1 = 100) \text{ AND } (COL2 < 12)) \text{ OR } ((COL1 = 100) \text{ AND } (COL2 = 12) \text{ AND } (COL3 < 99)))$

To define segment boundaries by all of the table partitions:



Only select this method if you are sure that the table is already partitioned.

1. In the **Parallel Load** tab's **Select Parallel Load Method** section, select **Use Partitions**.
2. In the **Select Partitions** section, select **Use all table partitions**. This will segment the table according to partitions that already exist in the source database.
3. Select one of the following:
 - Use main partitions
 - Use sub partitions



This option will be disabled if the source database does not support sub-partitions.

4. Click **OK**.

To define segment boundaries by specific partitions:



Only select this method if you are sure that the table is already partitioned.

1. In the **Parallel Load** tab's **Select Parallel Load Method** section, select **Use Partitions**.
2. In the **Select Partitions** section, select **Specify partitions**. This will split the data according to the specified source partitions.



*When **Specify partitions** is selected, only the specified partitions will be replicated.*

3. Click **Add Partition**.
4. Specify the name of an existing partition or sub-partition.
5. If you specified the name of a sub-partition, select the check box in the **Sub-Partition** column.



The check box will be disabled if the source database does not support sub-partitions.

6. Add additional partitions/sub-partitions as required.
7. To delete a partition/sub-partition, select the partition/sub-partition and then click **Delete**.
8. Click **OK** to save your settings.

Adjusting the number of segments that can be loaded in parallel

You can increase or decrease the number of segments that will be loaded in parallel. For example, if you selected the **Use all table partitions** option and the source table has 20 partitions, increasing the default number of concurrent tasks (5) may improve performance.



Sub-tasks are allocated for each segment \ partition \ sub partition.

*For example: If you select a table with 6 partitions and load the table using the **Use Partitions** method, 5 partitions will be loaded in parallel, corresponding with the default number of concurrent tasks (5). When one of the sub-tasks completes its run, it will be assigned to loading the sixth partition.*

The currently set value is displayed at the bottom of the **Parallel Load** tab. You can modify this value in the **Maximum number of tables to load in parallel** field in the [Full Load Tuning \(page 189\)](#) tab.

Handling LOB columns

You can override the [task's LOB settings](#) for individual tables.



This option is only available for tasks defined with any combination of the following source and target endpoints: Oracle source, Oracle target, PostgreSQL source, PostgreSQL target, Microsoft SQL Server source, Microsoft SQL Server target, MySQL source, and MySQL target.



- *During CDC or during Full Load when the **Allow unlimited LOB size** option is enabled, LOB data types are supported only in tables with a primary key or unique index.*
- *When replicating from Microsoft SQL Server, inline LOBS will always be read directly from the logs (i.e. without lookup).*

The following LOB handling options are available:

LOB handling options

Option	Description
Replicate LOB columns	<p>When this option is selected (the default), LOB columns will be replicated.</p> <p>Note that replicating LOBs may impact performance. This is especially true in the case of the large LOBs which require Replicate to perform a lookup from the source table in order to retrieve the source LOB value.</p> <div data-bbox="371 526 1390 842" style="border: 1px solid #ccc; padding: 10px; margin-top: 10px;"> <p> Enabling <i>Replicate LOB columns</i> while a task is running will not be reflected in the target tables, unless the target tables are recreated by Qlik Replicate. This should be done as follows:</p> <ol style="list-style-type: none"> a. Make sure that DROP and CREATE change table in the task settings' Store Changes Settings tab is selected (the default). b. Enable Replicate LOB columns and then reload the target tables. </div>
Allow unlimited LOB size	<p>Select this option - also known as Full LOB mode - to ensure that all LOBs are replicated without being truncated. This option should be selected when all (or nearly all) of the LOBs you wish to replicate are large (i.e. exceed 1 GB).</p> <div data-bbox="371 1003 1390 1171" style="border: 1px solid #ccc; padding: 10px; margin-top: 10px;"> <p> Note If the task's Change Processing Mode is set to "Batch optimized apply" (the default), Replicate will switch to "Transactional apply" mode to apply tables with LOBs.</p> </div>

Option	Description
Optimize handling when LOB size is less than (KB)	<p>Select this option when you need to replicate both small and large LOBs, and most of the LOBs are small.</p> <div data-bbox="376 389 1388 667" style="border: 1px solid #ccc; padding: 10px; margin: 10px 0;"> <p> <i>This option is supported with the following endpoints only:</i></p> <ul style="list-style-type: none"> • Sources: Oracle, Microsoft SQL server, MySQL, PostgreSQL, IBM DB2 for LUW, and Sybase ASE. • Targets: Oracle, Microsoft SQL Server, MySQL, PostgreSQL, IBM DB2 for z/OS, and Sybase ASE. </div> <p>When this option is selected, during Full Load, the small LOBs will be replicated "inline" (which is more efficient), and the large LOBs will be replicated by performing a lookup from the source table.</p> <p>When capturing changes, however, both small and large LOBs will be replicated by performing a lookup from the source table.</p> <div data-bbox="376 936 1388 1205" style="border: 1px solid #ccc; padding: 10px; margin: 10px 0;"> <p> <i>When this option is selected, Replicate will check all of the LOB sizes to determine which ones to transfer "inline". LOBs larger than the specified size will be replicated using Full LOB mode.</i></p> <p><i>Therefore, if you know that most of the LOBs are larger than the specified setting, it is better to use the Allow unlimited LOB size option instead.</i></p> </div>
Chunk size (KB)	<p>Optionally, change the size of the LOB chunks to use when replicating the data to the target. The default chunk size should suffice in most cases, but if you encounter performance issues, adjusting the size may improve performance.</p> <div data-bbox="376 1368 1388 1570" style="border: 1px solid #ccc; padding: 10px; margin: 10px 0;"> <p> <i>With some databases, data type validation occurs when the data is inserted or updated. In such cases, replication of structured data types (e.g. XML, JSON, GEOGRAPHY, etc.) may fail if the data is bigger than the specified chunk size.</i></p> </div>

Option	Description
Limit LOB size to (KB)	<p>Select this option if you only need to replicate small LOBs or if the target endpoint does not support unlimited LOB size. The maximum permitted value for this field is 102400 KB (100 MB).</p> <p>When replicating small LOBs, this option is more efficient than the Allow unlimited LOB size option since the LOBs are replicated "inline" as opposed to via "lookup" from the source. During Change Processing, small LOBs are usually replicated via "lookup" from the source.</p> <p>The limit applies to the number of LOB bytes that is read from the source endpoint. For BLOB columns, the target BLOB size will be exactly according to the specified limit. For CLOB and NCLOB columns, the target LOB size might differ from the specified limit if the source and target LOB do not have the same character set. In this case, Replicate will perform character set conversion, which might result in a discrepancy between the source and target LOB sizes.</p> <p>As the value of the Limit LOB size to is in bytes, the size should be calculated according to the following formulas:</p> <ul style="list-style-type: none"> • BLOB - The length of the largest LOB. • NCLOB - The length of the longest TEXT <i>in characters</i> multiplied by two (as each character is handled as a double-byte). If the data includes 4-byte characters, multiply it by four. • CLOB - The length of the longest TEXT in characters (as each character is handled as a UTF8 character). If the data includes 4-byte characters, multiply it by two. <div style="border: 1px solid #ccc; padding: 10px; margin-top: 10px;"> <p> • <i>Any LOBs larger than the specified size will be truncated.</i></p> <p>• <i>When capturing changes from Oracle source, inline BLOBs are replicated inline.</i></p> <p>• <i>Changes to this setting will only affect existing tables after they are reloaded.</i></p> </div>



In some scenarios, tasks configured to replicate tables with multiple LOB columns may consume a large amount of memory. This is because Replicate allocates memory by multiplying the **Limit LOB size to** value by the [Commit rate during full load](#) value, the sum of which, it multiplies by the number of LOB columns being replicated. So, for example, if LOB size is limited to 5 MB and the default commit rate is used (10000 events), a task replicating 6 LOB columns will consume 300 GB of memory. Note that other factors such as the database type and version may also affect memory consumption.

Should you encounter memory consumption issues and suspect that a combination of the above factors may be the cause, stop the task and lower the value in the [Commit rate during full load](#) field. Then resume the task. Repeat this process until acceptable performance/memory levels are reached.

These instructions apply to Change Processing and Full Load tasks.



Changes to a column's LOB size while a task is running will not be reflected in the Change Table, unless the target tables are created by Qlik Replicate. In such cases, the task must be configured to drop and create the Change Table (the default) and the target tables need to be reloaded (after the LOB size has changed).

For more information on the Change Table, see [Store Changes Settings \(page 191\)](#). For information on reloading target tables, see the [Qlik Replicate Help](#).

Message format

This tab is only available for tasks defined with a supported streaming endpoint.

When a task is defined with such an endpoint, you can specify a custom message format that will override the default Replicate message format. This may be useful if the consumer application needs to process the message in a particular format.

The custom message format can be defined at task level and/or at table level. When it is defined at both task *and* table level, the message format defined for the table will take precedence over the message format defined for the task.

To define a custom message at table level:

1. Select a table.
2. Open the **Table Settings** window as described in [Table Settings \(page 91\)](#).
3. Select the **Message Format** tab and click the **Change to Table Policy** button.
4. Configure the message format as described in [Message format \(page 214\)](#).
5. To use the message format defined for the task, click the **Change to Task Policy** button.

For information on defining a custom message at task level, see [Message format \(page 214\)](#).

Full Load

This tab is available for tasks defined with the IBM DB2 for z/OS and IBM DB2 for iSeries source endpoints only.

Select the **Eliminate creation of duplicate records on full load** option if you need to prevent duplicate records from being replicated during Full Load. You can either set the option at task level or per table.

Note that selecting this option could impact performance as Replicate instructs the source database to return the table records by Primary Key order and then removes any duplicate records.

To prevent creation of duplicate records per table:

1. Select the desired table and then open the **Table Settings** window as described in [Table Settings \(page 91\)](#).
2. Select the **Full Load** tab and click the **Change to Table Policy** button.
3. Select the **Prevent creation of duplicate records on the target** check box.

For information on preventing creation of duplicate records at task level, see [Full Load Settings \(page 187\)](#).

9.2 Defining global rules

Global rules are a useful way of making changes across multiple tables and columns in the same task. You can define *transformation rules* that modify the source data or metadata before it reaches the target, and/or you can define *filter rules* that determine which records will be replicated to the target.



Global rules are not available in a Log Stream Staging setup.

For information on Log Stream Staging, refer to the Qlik Replicate Setup and User Guide.

- **Transformations** - One way transformations can be used is to change the names of all tables in a task. You can change the names using wildcards and patterns. For example, you may want to change the names of the tables from `account_%` to `ac_%`. This is helpful when replicating data from a Microsoft SQL Server endpoint to an Oracle endpoint where the Microsoft SQL Server endpoint has a limit of 128 characters for a table name and the Oracle endpoint has a limit of 31 characters. You may also need to change a specific data type in the source to a different data type in the target for many or all of the tables in the task. Global transformation will allow you to accomplish this without having to define a transformation for each individual table.



Table-specific transformations override global transformations. For example, you can define a global transformation that changes the data type for all tables from `DATE` to `DATETIME(6)` and then define another transformation for a specific table that changes the data type from `DATE` to `STRING(50)`.

For information on defining a transformation for a specific table, see [Defining Transformations for a Single Table/View](#).

For an explanation of how to create global transformation, see [Starting the Global Transformation Rules wizard \(page 120\)](#).

- **Filters** - Use filter rules to determine which records will be replicated to the target. Filter can be based on column data (e.g. only replicate records where Age is greater than 18) or record attributes (e.g. only replicate UPDATED records).

For an explanation of how to create global filters, see [Starting the Global Filter Rules wizard \(page 144\)](#).

Starting the Global Transformation Rules wizard

You define global transformations using the Global Transformation Rules wizard.

To start the Global Transformations wizard:

1. Open the task for which you want to create a global transformations or a global filter.
You can click **View Task** above the Tasks list or double-click the task.
2. If you are not in the Designer mode, click **Designer** at the top right of the screen.
For more information on the Designer mode, see [Designer mode \(page 225\)](#).
3. In Designer mode, click **Global Rules**.
The [Global Rules](#) window opens.
4. Click the **New Rule** toolbar button and select **Transformation**.
The **New Transformation Rule** wizard opens.
5. Enter the information to define a global transformation rule. The first step is selecting the [Transformation type \(page 120\)](#).

Limitations for global transformations

The following limitations apply to global transformations:

- Transformations are not supported for columns with Right-to-Left languages.
- By default, transformations cannot be performed on columns that contain special characters (e.g. #, \, /, -) in their name.
However, such functionality is provided via the **Support special characters in column names used in expressions** in the [task settings](#). If you intend to enable option, make sure to carefully review the requirements first.
- The only supported transformation for columns that are mapped to BLOB/CLOB data types (by Replicate) is to drop the column on the target.
- Expressions must be written using SQLite syntax only.
- Changing a global transformation value will not reload affected tables automatically. The target must be reloaded manually for the changes to take effect.
For information on reloading the target, see [Using the Run button options \(page 253\)](#).

Transformation type

In the **Transformation type** step of the **New Transformation Rule** wizard, you define the type of transformation you want to be performed.



You can only create one rule for each transformation type on the same object (e.g. a column). If you create multiple rules for a single transformation type on the same object, only the last rule you create will be valid. For example, if you create the following rules (in order) to rename a schema:

Rename Schema: Add Prefix

Rename Schema: Add Suffix

OR

Rename Column: Add Prefix

Rename Column: Add Suffix

Only the second rule (adding a suffix) will be executed.

To select the transformation type:

1. Enter a name for the rule.
The name cannot exceed 32 characters, contain non-Latin characters, or contain any of the following characters: \:*?"<>|

2. Select one of the following:

Table or Schema:

- **Rename schema:** Select this to change the schema name for multiple tables. For example, if you want all HR tables to be renamed PERS.
- **Rename table:** Select this to change the name of multiple tables. For example, if you want all tables named SALARY to be called WAGES.

Tablespace:

- **Change table tablespace:** Select this to change the table tablespace on the target. You can change the table tablespace regardless of what objects it contains or you can specify a condition for it to be renamed. For example, change all table tablespaces that contain the table EMPLOYEES in the schema COMPANY.
By default (i.e. when this option is not selected), the tables will either be created in the source table tablespace on the target (when replicating from an Oracle source) or in the default database tablespace (when replicating from any other source).



This option is only available for tasks with an Oracle target endpoint.

- **Change index tablespace:** Select this to change the index tablespace on the target. You can change the index tablespace regardless of what objects it contains or you can specify a condition for it to be renamed. For example, change all table tablespaces that contain the table EMPLOYEES in the schema COMPANY.

By default (i.e. when this option is not selected), the indexes will either be created in the source table tablespace on the target (when replicating from an Oracle source) or in the default database tablespace (when replicating from any other source).



This option is only available for tasks with an Oracle target endpoint.

Column:

- **Rename column:** Select this to change the name of multiple columns. For example, if you want to change all columns with word `MINIMUM` to `MIN`.
- **Add column:** Select this to add a column with a similar name to multiple tables.
- **Drop column:** Select this to drop a column with a similar name from multiple tables.
- **Convert data type:** Select this if you want to change a specific data type to a different one across multiple tables. For example, if you want to change all Integer data types to a string.
- **Replace column data:** Select this to replace column data across multiple target tables.



In homogeneous replication tasks (such as Oracle to Oracle), modifying a single table column (by changing the column data type or length for example), will break the homogeneity for the entire table.

Change Table:

Change Table transformations are only available when the [Store Changes](#) replication option is enabled.

For more information on Change Tables, refer to the *Qlik Replicate Setup and User Guide*.

- **Rename Change Table:** Select this to rename the Replicate Change Table for all tables or for any table that matches the specified schema name and/or table name.
- **Rename Change Table schema:** Select this to change the schema under which the Replicate Change Table will be created, for all tables or for any table that matches the specified schema name and/or table name.

3. Click **Next** to proceed to the [Transformation scope \(page 122\)](#) step.

Transformation scope

In the **Transformation scope** screen, you define which tables will be affected by the transformation. For example, you can apply the rule to all tables that contain the word `SALARY` as part of their name.



The options displayed in this screen depend on the selected [Transformation Type](#).

The following table describes all available options. The second column lists the Transformation Type where the option to be available.

Transformation conditions

Option	Transformation Type	Description
Schema name is like %	All	<p>Leave the % sign to include all schemas in your global transformation.</p> <p>Click the % sign to add a filter. In this case you can enter any name combination to include only that schema in your global transformation rule.</p> <p>For example, enter HR to include only tables that have the schema HR.</p> <p>You can use the % sign as a wildcard. For example, H% includes all tables with a schema that begins with the letter H, such as HR, HELLO, or HQ.</p> <p>The % wildcard can be used in any position. For example, if you use it at the beginning, %H, then all table names that end in H are included in the transformation rule. The % can also be used in a middle position.</p> <div data-bbox="770 1014 1390 1494" style="border: 1px solid gray; padding: 10px; margin-top: 10px;"> <p> <i>If you are using an Oracle target, you must enter a schema that exists on the target endpoint. Qlik Replicate does not create new schemas on an Oracle endpoint. If you want to use a new schema for the target, create the schema on the Oracle endpoint before running the task. For more information, see the topic "Configuring an Oracle database as a Qlik Replicate Target" in the Qlik Replicate User and Reference Guide.</i></p> </div>

Option	Transformation Type	Description
<p>Table tablespace is like %</p>	<p>Change table tablespace</p>	<div data-bbox="772 293 1390 427" style="border: 1px solid #ccc; padding: 5px; margin-bottom: 10px;">  <i>This option is only available if the task is defined with an Oracle target endpoint.</i> </div> <p>Leave the % sign to include all table tablespace names in your global transformation.</p> <p>Click the % sign to add a filter. In this case, you can enter any name combination to include only the specified table tablespace in your global transformation rule.</p> <p>For example, enter <code>SYSTEM</code> to include only table tablespaces called <code>SYSTEM</code>.</p> <p>You can also use the % sign as a wildcard anywhere in the string. For example, <code>H%</code> includes all table tablespaces that begin with the letter "H" whereas <code>%H</code> includes all table tablespaces that end with the letter "H".</p>
<p>Index tablespace is like %</p>	<p>Change index tablespace</p>	<div data-bbox="772 1041 1390 1176" style="border: 1px solid #ccc; padding: 5px; margin-bottom: 10px;">  <i>This option is only available if the task is defined with an Oracle target endpoint.</i> </div> <p>Leave the % sign to include all index tablespace names in your global transformation.</p> <p>Click the % sign to add a filter. In this case, you can enter any name combination to include only the specified index tablespace in your global transformation rule.</p> <p>For example, enter <code>SYSTEM</code> to include only index tablespaces called <code>SYSTEM</code>.</p> <p>You can also use the % sign as a wildcard anywhere in the string. For example, <code>H%</code> includes all index tablespaces that begin with the letter "H" whereas <code>%H</code> includes all index tablespaces that end with the letter "H".</p>

Option	Transformation Type	Description
Table name is like %	All	<p>Leave the % sign to include all table names in your global transformation rule.</p> <p>Click the % sign to add a filter. In this case you can enter any name combination to include only tables with that specific name in your global transformation rule.</p> <p>You can use the % sign as a wildcard. For example, J% includes all tables with a name that begins with the letter J, such as JOBS, JOBS_HISTORY, or JACKSONVILLE.</p> <p>The % wildcard can be used in any position. For example, if you use it at the beginning, %H, then all table names that end in H are included in the transformation rule. The % can also be used in a middle position.</p>
Column name is like %	Rename column Drop column Convert data type Replace column value	<p>Leave the % sign to include all column names in your global transformation rule.</p> <p>Click the % sign to add a filter. In this case you can enter any name combination to include only columns with that specific name in your global transformation rule.</p> <p>You can use the % sign as a wildcard. For example, N% includes all columns with a name that begins with the letter N, such as NAME, NAME_FIRST, or NAME_LAST.</p> <p>The % wildcard can be used in any position. For example, if you use it at the beginning, %IES, then all column names that end in with the string "IES" are included in the transformation rule. The % can also be used in a middle position.</p>
Data type is	Convert data type Replace column value	<p>Select a new data type from the drop-down list. Make sure that the data type you select is compatible with the data in that column.</p> <p>For a description of Qlik Replicate data types, information about data type mapping from the native endpoint to Qlik Replicate, or for a list of endpoints supported by Qlik Replicate, see the Qlik Replicate User and Reference Guide.</p>
Scope expression	All	Click Advanced options to define a scope expression using the Expression Builder.

After you complete defining the transformation rule definitions, click **Next** to go to the [Transformation action \(page 126\)](#) step.



*If the global transformation type you are defining is **Drop Column**, you do not need to create a [Transformation Rule](#). In this case, click **Finish** to add the rule to the [Global Rules](#) list.*

Transformation action

In the **Transformation action** screen, you define what happens to the objects affected by the transformation rule. For example, you can define a new name for the affected objects or add a prefix to the table names. Only objects that fall within the [Transformation scope \(page 122\)](#) will be affected.

The following transformation options are available:

- [Rename Schema \(page 127\)](#)
- [\(page 130\)](#)
- [\(page 130\)](#)
- [Rename Table \(page 130\)](#)
- [Rename Column \(page 133\)](#)
- [Add column \(page 136\)](#)
- [Drop Column \(page 136\)](#)
- [Convert data type \(page 136\)](#)
- [Rename Change Table schema \(page 141\)](#)
- [Rename Change Table \(page 138\)](#)

When done, click **Next**.

Limitations for transformation rules

The following limitations apply to transformation rules:

- Transformations are not supported for columns with Right-to-Left languages.
- By default, transformations cannot be performed on columns that contain special characters (e.g. #, \, /, -) in their name.
However, such functionality is provided via the **Support special characters in column names used in expressions** in the [task settings](#). If you intend to enable option, make sure to carefully review the requirements first.
- The only supported transformation for columns that are mapped to BLOB/CLOB data types (by Replicate) is to drop the column on the target.
- Expressions must be written using SQLite syntax only.
- Changing a global transformation value will not reload affected tables automatically. The target must be reloaded manually for the changes to take effect.
For information on reloading the target, see [Using the Run button options \(page 253\)](#).



The options displayed in this screen depend on the [Transformation Type](#) selected.

Rename Schema

If your transformation type is **Rename Schema**, you can do the following:

- [Rename schema to \(string\) \(page 127\)](#)
- [Add a prefix or suffix \(page 127\)](#)
- [Remove a prefix or suffix \(page 127\)](#)
- [Replace a prefix or suffix with different characters \(page 128\)](#)
- [Convert schema name to uppercase \(page 129\)](#)
- [Convert schema name to lowercase \(page 129\)](#)
- [Rename schema \(Expression\) \(page 129\)](#)

Rename schema to (string)

Use the **Rename schema to: [string]** option to change the name of all table schemas that you defined in the [Transformation scope \(page 122\)](#) step to a different name. For example, if you have a schema called `Human_Resources` and want to change all instances of this name to `HR` then enter the string `HR`. You can enter any string in this field.

Add a prefix or suffix

Use the **Add a prefix or suffix** option to add additional characters to the beginning or end of the schema name for all schemas that fit the definition you created in the [Transformation scope \(page 122\)](#) step. For example, if the schema name is `HR`, you can add a suffix, such as `TAR` or `_TAR` to the schema name for all tables with that schema name. In this case, the resulting schema name will be `HRTAR` or `HR_TAR`.



If you are using Oracle as your target endpoint, Qlik Replicate does not create a new schema. Therefore, the schema name that is the result of replacing a prefix or suffix with a different string of characters must exist in the Oracle target endpoint. If the resulting schema name does not exist, you must create the schema in the Oracle endpoint before carrying out this task.

For more information, see the Qlik Replicate Setup and User Guide.

To globally add a prefix or suffix

1. Select **Add <Prefix/Suffix> Insert Characters to matching schema names**.
2. Click the word **Prefix** or **Suffix** and select one of these two from the list.
3. Click **[string]** to activate the field.
4. Type the characters you want as the prefix or suffix. If you want to include an underscore or other legal character to separate the prefix/suffix from the original name, you must add it as part of the character string.
5. Click **Finish** to add the rule to the [Global Rules](#) list.

Remove a prefix or suffix

Use the **Remove a prefix or suffix** option to remove a string of characters from the beginning or end of a schema name for all schema that fit the definition you created in the [Transformation scope \(page 122\)](#) step.

For example, you can use this option to remove the letters `_REV` from the schema name for all tables in the schema `HR_REV`. In this case the schema name in the target will be `HR`.



If you are using Oracle as your target endpoint, Qlik Replicate does not create a new schema. Therefore, the schema name that is the result of replacing a prefix or suffix with a different string of characters must exist in the Oracle target endpoint. If the resulting schema name does not exist, you must create the schema in the Oracle endpoint before carrying out this task.

For more information, see the Qlik Replicate Setup and User Guide.

To globally remove a prefix or suffix

1. Select **Remove <Prefix/Suffix> Insert Characters from matching schema names**.
2. Click the word **Prefix** or **Suffix** and select one of these two from the list.
3. Click **[string]** to activate the field.
4. Type the characters you want to remove. If you want to remove an underscore or other legal character from the original name, you must add it as part of the character string.
5. Click **Finish** to add the rule to the [Global Rules](#) list.

Replace a prefix or suffix with different characters

Use the **Replace a prefix or suffix** option to replace a string of characters with a different string of characters. You determine whether to replace the characters at the beginning or end of a schema name for all schema that fit the definition you created in the [Transformation scope \(page 122\)](#) step.

For example, you can use this option to replace the letters `_ORIG` with `_REPL` in the schema name for all tables in the schema `HR_ORIG`. In this case the schema name in the target will be `HR_REPL`.



If you are using Oracle as your target endpoint, Qlik Replicate does not create a new schema. Therefore, the schema name that is the result of replacing a prefix or suffix with a different string of characters must exist in the Oracle target endpoint. If the resulting schema name does not exist, you must create the schema in the Oracle endpoint before carrying out this task.

For more information, see the Qlik Replicate Setup and User Guide.

To globally replace a prefix or suffix

1. Select **Replace <Prefix/Suffix> Insert Characters by Insert Characters for all matching schema names**.
2. Click the word **Prefix** or **Suffix** and select one of these two from the list.
3. Click the first **[string]** to activate the field.
4. Type the characters from the existing (source) schema that you want to replace. If you want to include an underscore or other legal character from the original name in the string that you want to replace, you must add it as part of the character string.
5. Click the second **[string]** to activate the field.

6. Type the characters you want to use in the target. These characters replace the original (source) characters in the target.
7. Click **Finish** to add the rule to the [Global Rules](#) list.

Convert schema name to uppercase

Use the convert to uppercase option to convert all of the letters in a schema name to upper case. For example:

- Schema_cat, becomes SCHEMA_CAT
- schema_cat, becomes SCHEMA_CAT
- sChEMa_Cat, becomes SCHEMA_CAT

To globally change the schema name to all uppercase

1. Select **Convert schema name to uppercase**.
2. Click **Finish** to add the rule to the [Global Rules](#) list.

Convert schema name to lowercase

Use the convert to lowercase option to convert all of the letters in a schema name to lower case. For example:

- Schema_cat, becomes schema_cat
- SCHEMA_CAT, becomes schema_cat
- sChEMa_Cat, becomes schema_cat

To globally change the schema name to all lowercase

1. Select **Convert schema name to lowercase**.
2. Click **Finish** to add the rule to the [Global Rules](#) list.

Rename schema (Expression)

Use the **Rename schema to [expression]** option to change the name of all table schemas that you defined in the [Transformation scope \(page 122\)](#) step to a different name. For example, if you have a schema called Human_Resources and want to change all instances of this name to HR.



If you are using Oracle as your target endpoint, Qlik Replicate does not create a new schema. Therefore, the schema name that is the result of replacing a prefix or suffix with a different string of characters must exist in the Oracle target endpoint. If the resulting schema name does not exist, you must create the schema in the Oracle endpoint before carrying out this task.

For more information, see the Qlik Replicate Setup and User Guide.

To globally change a schema name

1. Select **Rename schema to [expression]**
2. Click the button to the right of the **Rename schema** option to open the Expression Editor. For information on how to use the Expression Editor, see [Using the Expression Builder \(page 149\)](#). Then go to step 4.
or
Click **[expression]** to activate the field and continue with step 3.
3. Type an SQLite expression or a string (in quotes) to rename the schema. For example:
 - "New_Schema"
 - 'PREFIX_||\$SCHEMA_NAME_VAR||'_SUFF'You can use the following variables in the SQLite expression:
 - \$SCHEMA_NAME_VAR
 - \$TABLE_NAME_VAR
 - \$COLUMN_NAME_VAR
 - \$COLUMN_DATATYPE_VAR
4. Click **Finish** to add the rule to the [Global Rules](#) list.

Change table tablespace

If your transformation type is **Change table tablespace**, you can change the table tablespace on an Oracle target. You can also set certain conditions that must exist in the source for the table tablespace to be changed. These include schema name, table name and table tablespace name.

For more information, see the following topics:

- [Transformation type \(page 120\)](#)
- [Transformation action \(page 126\)](#)

Change index tablespace

If your transformation type is **Change index tablespace**, you can change the index tablespace on an Oracle target. You can also set certain conditions that must exist in the source for the tablespace to be changed. These include schema name, table name and index tablespace name.

For more information, see the following topics:

- [Transformation type \(page 120\)](#)
- [Transformation action \(page 126\)](#)

Rename Table

If your transformation type is **Rename Table**, you can do the following:

- [Rename table to \(string\) \(page 131\)](#)
- [Add a prefix or suffix \(page 131\)](#)
- [Remove a prefix or suffix \(page 131\)](#)

- [Replace a prefix or suffix with different characters \(page 132\)](#)
- [Convert table name to uppercase \(page 132\)](#)
- [Convert table name to lowercase \(page 132\)](#)
- [Rename table \(expression\) \(page 133\)](#)

Rename table to (string)

Use the **Rename table to: [string]** option to change the name of all tables that you defined in the [Transformation scope \(page 122\)](#) step to a different name. For example, if you have a table called EMPLOYEE and want to change all instances of this name to EMP then enter the string EMP. You can enter any string in this field.

Add a prefix or suffix

Use the **Add a prefix or suffix** option to add additional characters to the beginning or end of the table name for all tables that fit the definition you created in the [Transformation scope \(page 122\)](#) step. For example, if the table name is EMPLOYEES, you can add a suffix, such as TAR or _TAR to the table name for all tables with that table name. In this case, the resulting table name will be EMPLOYEEESTAR or EMPLOYEES_TAR.

To globally add a prefix or suffix:

1. Select **Add <Prefix/Suffix> Insert Characters to matching table names**.
2. Click the word **Prefix** or **Suffix** and select one of these two from the list.
3. Click **[string]** to activate the field.
4. Type the characters you want as the prefix or suffix. If you want to include an underscore or other legal character to separate the prefix/suffix from the original name, you must add it as part of the character string.
5. Click **Finish** to add the rule to the [Global Rules](#) list.

Remove a prefix or suffix

Use the **Remove a prefix or suffix** option to remove a string of characters from the beginning or end of a table name for all tables that fit the definition you created in the [Transformation scope \(page 122\)](#) step.

For example, you can use this option to remove the letters _REV from the table name for all tables with the name EMPLOYEES. In this case the table name in the target will be EMPLOYEES.

To globally remove a prefix or suffix:

1. Select **Remove <Prefix/Suffix> Insert Characters from matching table names**.
2. Click the word **Prefix** or **Suffix** and select one of these two from the list.
3. Click **[string]** to activate the field.
4. Type the characters you want to remove. If you want to remove an underscore or other legal character from the original name, you must add it as part of the character string.
5. Click **Finish** to add the rule to the [Global Rules](#) list.

Replace a prefix or suffix with different characters

Use the **Replace a prefix or suffix** option to replace a string of characters with a different string of characters. You determine whether to replace the characters at the beginning or end of a table name for all tables that fit the definition you created in the [Transformation scope \(page 122\)](#) step.

For example, you can use this option to replace the letters `_ORIG` with `_REPL` in the table names for all tables called `EMPLOYEE_ORIG`. In this case the table name in the target will be `EMPLOYEE_REPL`.

To globally replace a prefix or suffix:

1. Select **Replace <Prefix/Suffix> Insert Characters by Insert Characters for all matching schema names**.
2. Click the word **Prefix** or **Suffix** and select one of these two from the list.
3. Click the first **[string]** to activate the field.
4. Type the characters from the existing (source) schema that you want to replace. If you want to include an underscore or other legal character from the original name in the string that you want to replace, you must add it as part of the character string.
5. Click the second **[string]** to activate the field.
6. Type the characters you want to use in the target. These characters replace the original (source) characters in the target.
7. Click **Finish** to add the rule to the [Global Rules](#) list.

Convert table name to uppercase

Use the convert to uppercase option to convert a table name to all upper case. For example:

- `Table_cat`, becomes `TABLE_CAT`
- `table_cat`, becomes `TABLE_CAT`
- `taBLe_Cat`, becomes `TABLE_CAT`

To globally change the table name to all uppercase:

1. Select **Convert table name to uppercase**.
2. Click **Finish** to add the rule to the [Global Rules](#) list.

Convert table name to lowercase

Use the convert to lowercase option to convert a table name to all lower case. For example:

- `Table_cat`, becomes `table_cat`
- `TABLE_CAT`, becomes `table_cat`
- `taBLe_Cat`, becomes `table_cat`

To globally change the table name to all lowercase:

1. Select **Convert table name to lowercase**.
2. Click **Finish** to add the rule to the [Global Rules](#) list.

Rename table (expression)

Use the **Rename table to [expression]** option to change the name of all tables that fit the definition you created in the [Transformation scope \(page 122\)](#) step. For example, if you have a table called EMPLOYEE and want to change all instances of this name as defined in the previous step it to EMP.

To change the table name:

1. Select **Rename table to: [expression]**
2. Click the button to the right of the **Rename table** option to open the Expression Editor. For information on how to use the Expression Editor, see [Using the Expression Builder \(page 149\)](#). Then go to step 4.
or
Click **[expression]** to activate the field and continue with step 3.
3. Type an SQLite expression or a string (in quotes) to rename the table. For example:

- "New_Table"
- 'PREFIX_'||\$TABLE_NAME_VAR||'_SUFFIX'

You can use the following variables in the SQLite expression:

- \$SCHEMA_NAME_VAR
- \$TABLE_NAME_VAR
- \$COLUMN_NAME_VAR
- \$COLUMN_DATATYPE_VAR

Rename Column

If your transformation type is **Rename Column**, you can do the following:

- [Rename column to \(string\) \(page 133\)](#)
- [Add a prefix or suffix \(page 133\)](#)
- [Remove a prefix or suffix \(page 134\)](#)
- [Replace a prefix or suffix with different characters \(page 134\)](#)
- [Convert column name to uppercase \(page 135\)](#)
- [Convert column name to lowercase \(page 135\)](#)
- [Rename column \(expression\) \(page 135\)](#)

Rename column to (string)

Use the **Rename column to: [string]** option to change the name of all columns that you defined in the [Transformation scope \(page 122\)](#) step to a different name. For example, if you have a table called SALARY and want to change all instances of this name to EMP then enter the string SAL. You can enter any string in this field.

Add a prefix or suffix

Use the **Add a prefix or suffix** option to add additional characters to the beginning or end of the column name for all columns that fit the definition you created in the [Transformation scope \(page 122\)](#) step. For example, if the column name is SALARY, you can add a suffix, such as TAR or _TAR to the table name for all tables with that table name. In this case, the resulting table name will be SALARYTAR or SALARY_TAR.

To globally add a prefix or suffix:

1. Select **Add <Prefix/Suffix> Insert Characters to matching column names**.
2. Click the word **Prefix** or **Suffix** and select one of these two from the list.
3. Click the **[string]** to activate the field.
4. Type the characters you want as the prefix or suffix. If you want to include an underscore or other legal character to separate the prefix/suffix from the original name, you must add it as part of the character string.
5. Click **Finish** to add the rule to the [Global Rules](#) list.

Remove a prefix or suffix

Use the **Remove a prefix or suffix** option to remove a string of characters from the beginning or end of a column name for all columns that fit the definition you created in the [Transformation scope \(page 122\)](#) step.

For example, you can use this option to remove the letters `_REV` from the column name for all columns with the name `SALARY`. In this case the column name in the target will be `SALARY`.

To globally remove a prefix or suffix:

1. Select **Remove <Prefix/Suffix> Insert Characters from matching column names**.
2. Click the word **Prefix** or **Suffix** and select one of these two from the list.
3. Click **[string]** to activate the field.
4. Type the characters you want to remove. If you want to remove an underscore or other legal character from the original name, you must add it as part of the character string.
5. Click **Finish** to add the rule to the [Global Rules](#) list.

Replace a prefix or suffix with different characters

Use the **Replace a prefix or suffix** option to replace a string of characters with a different string of characters. You determine whether to replace the characters at the beginning or end of a column name for all columns that fit the definition you created in the [Transformation scope \(page 122\)](#) step.

For example, you can use this option to replace the letters `_ORIG` with `_REPL` in the column names for all columns called `SALARY_ORIG`. In this case the column name in the target will be `SALARY_REPL`.

To globally replace a prefix or suffix:

1. Select **Replace <Prefix/Suffix> Insert Characters by Insert Characters for all matching schema names**.
2. Click the word **Prefix** or **Suffix** and select one of these two from the list.
3. Click the first **[string]** to activate the field.
4. Type the characters from the existing (source) column that you want to replace. If you want to include an underscore or other legal character from the original name in the string that you want to replace, you must add it as part of the character string.
5. Click the second **[string]** to activate the field.
6. Type the characters you want to use in the target. These characters replace the original (source)

characters in the target.

7. Click **Finish** to add the rule to the [Global Rules](#) list.

Convert column name to uppercase

Use the convert to uppercase option to convert a column name to all upper case. For example:

- Column_cat, becomes COLUMN_CAT
- column_cat, becomes COLUMN_CAT
- coLUMnM_Cat, becomes COLUMN_CAT

To globally change the table name to all uppercase

1. Select **Convert column name to uppercase**.
2. Click **Finish** to add the rule to the [Global Rules](#) list.

Convert column name to lowercase

Use the convert to lowercase option to convert a column name to all lower case. For example:

- Column_cat, becomes column_cat
- column_cat, becomes column_cat
- coLUMnM_Cat, becomes column_cat

To globally change the column name to all lowercase:

1. Select **Convert column name to lowercase**.
2. Click **Finish** to add the rule to the [Global Rules](#) list.

Rename column (expression)

Use the **Rename column to [expression]** option to change the name of all tables that fit the definition you created in the [Transformation scope \(page 122\)](#) step. For instance, if you have a column called SALARY and want to change it to SAL.

To change the column name:

1. Select **Rename column to: [expression]**
2. Click the button to the right of the **Rename column** option to open the Expression Editor. For information on how to use the Expression Editor, see [Using the Expression Builder \(page 149\)](#). Then go to step 4.
or
Click **[expression]** to activate the field and continue with step 3.
3. Type an SQLite expression or a string (in quotes) to rename the column. For example:

- "New_Column"
- 'PREF_'||\$COLUMN_NAME_VAR||'_SUFF'

You can use the following variables in the SQLite expression:

- \$SCHEMA_NAME_VAR
- \$TABLE_NAME_VAR

- \$COLUMN_NAME_VAR
- \$COLUMN_DATATYPE_VAR

Add column

When you add a column to multiple tables, you must provide a name, define the data type for the column and define the data that the column contains. The column that you define here is added to all tables that fit the definition you created in step [Transformation scope \(page 122\)](#).

The following describes the information you must enter in the transformation rule page for adding a column.

- **Column name:** Click the **[string]** to activate the field. Type the name for the column in the field. A column with this name is added to all tables that fit the definition you created in step [Transformation scope \(page 122\)](#).
- **Add to Primary Key:** Select to add the column to the target tables' Primary Key.
- **Expression:** Click the button to the right of this field to open the Expression Builder or type an expression using SQLite operators to define the data in the column. For information on how to use the Expression Editor to create an expression, see [Using the Expression Builder \(page 149\)](#). For more information on creating expressions, see [Creating an expression for transformations \(page 102\)](#) and [Using SQLite syntax with transformations \(page 102\)](#).
- **Set target data type to:** Click the drop-down for a list of data types and select a new data type from the drop-down list. Make sure that the data type you select is compatible with the data in that column. When **BYTES**, **STRING** or **WSTRING** is selected, specify a **Length** as well. When **NUMERIC** is selected, specify a **Precision and Scale** as well.
- **Subtype:** When **CLOB**, **NCLOB**, **STRING**, or **WSTRING** data types are selected, you can also set a data subtype. Select either **JSON** or **XML** from the **Subtype** drop-down list. Make sure that the data in the new column will be compatible with the selected subtype. The default is **Regular**, which means that the regular data type will be used without a subtype.

For a description of the available data types and for a list of endpoints supported by Qlik Replicate, and for information about data type mapping from source endpoints to Qlik Replicate data types, see the [Qlik Replicate online help](#)

Drop Column

This option does not require a transformation rule. For this option you complete the Global transformation rule after the [Transformation scope \(page 122\)](#) step.

Convert data type

The data type that you define in this step is applied to all columns and tables that fit the definition you created in the [Transformation scope \(page 122\)](#) step. Make sure that the data type you select is compatible with the data in the columns you defined.

- **Set target data type to** - If you change the target value type (e.g. string to numeric), you may also need to change the data type of the target columns as well.

For the **BYTES**, **STRING**, and **WSTRING** data types, you can optionally specify the **Length** as well. If you leave the default value ("0"), Replicate will calculate the data type length based on the source column definition. You can also set the length using an expression. When you click the **fx** button to the right of the **Length** field, the Expression Builder opens showing the **Metadata** tab. For an explanation of the variables in this tab, see [Metadata \(Global transformations only\) \(page 155\)](#).

Example:

The following expression multiplies the modified data type length by two.

```
$AR_M_MODIFIED_DATATYPE_LENGTH * 2
```

For the **NUMERIC** data type, you can optionally set the **Precision** and **Scale**. If you leave the default value ("0"), Replicate will calculate the precision and/or scale based on the source column value.

- **Subtype:** When **CLOB**, **NCLOB**, **STRING**, or **WSTRING** data types are selected, you can also set a data subtype. Select either **JSON** or **XML** from the **Subtype** drop-down list. Make sure that the new data in the column will be compatible with the selected subtype. The default is **Regular**, which means that the regular data type will be used without a subtype.

For a description of the available data types and for a list of endpoints supported by Qlik Replicate, and for information about data type mapping from source endpoints to Qlik Replicate data types, see the [Qlik Replicate online help](#)

Replace column value

Use the **Replace column value** transformation to replace the values in the source columns (set in the **Transformation scope**) with different values in the corresponding target columns.

The following options are available:

- **Replace target value with** - Create an expression for replacing the value in the source column values with a different value in the target columns. When you click the **fx** button to the right of the field, the Expression Builder opens showing the **Data** tab. For an explanation of the variables in this tab, see [Data \(global transformations only\) \(page 154\)](#).

Example:

The following expression appends the string "_new" to the original source column values.

```
$AR_M_SOURCE_COLUMN_DATA || '_new'
```

- **Set target data type to** - If you change the target value type (e.g. string to numeric), you may also need to change the data type of the target columns as well. For the **BYTES**, **STRING**, and **WSTRING** data types, you can optionally specify the **Length** as well. If you leave the default value ("0"), Replicate will calculate the data type length based on the source column definition. You can also set the length using an expression. When you click the **fx** button to the right of the **Length** field, the Expression Builder opens showing the **Metadata** tab. For an explanation of the variables in this tab, see [Metadata \(Global transformations only\) \(page 155\)](#).

Example:

The following expression multiplies the modified data type length by two.

```
$AR_M_MODIFIED_DATATYPE_LENGTH * 2
```

For the **NUMERIC** data type, you can optionally set the **Precision** and **Scale**. If you leave the default value ("0"), Replicate will calculate the precision and/or scale based on the source column value.

- **Subtype:** When **CLOB**, **NCLOB**, **STRING**, or **WSTRING** data types are selected, you can also set a data subtype. Select either **JSON** or **XML** from the **Subtype** drop-down list. Make sure that the new data in the column will be compatible with the selected subtype. The default is **Regular**, which means that the regular data type will be used without a subtype.

See also: [Using the Expression Builder \(page 149\)](#)

Rename Change Table

This transformation is only available when the Store Changes replication option is turned on.

If your transformation type is **Rename Change Table**, you can do the following:

- [Rename Change Table to \(string\) \(page 138\)](#)
- [Add a prefix or suffix \(page 139\)](#)
- [Remove a prefix or suffix \(page 139\)](#)
- [Replace a prefix or suffix with different characters \(page 139\)](#)
- [Convert Change Table name to uppercase \(page 140\)](#)
- [Convert Change Table name to lowercase \(page 140\)](#)
- [Rename Change Table \(expression\) \(page 140\)](#)



- Globally renaming a Change Table will override the Change Table [suffix defined in the task settings](#).
- The Change Table name must be different from the source table names. Otherwise, a table error will occur.

Rename Change Table to (string)

Use the **Rename Change Table to: [string]** option to change the name of all Change Tables that you defined in the [Transformation scope \(page 122\)](#) step to a different name. For example, if you have a Change Table called `EMPLOYEE` and want to change all instances of this name to `EMP` then enter the string `EMP`. You can enter any string in this field.

Add a prefix or suffix

Use the **Add a prefix or suffix** option to add additional characters to the beginning or end of the Change Table name for all Change Tables that fit the definition you created in the [Transformation scope \(page 122\)](#) step. For example, if the Change Table name is `EMPLOYEES`, you can add a suffix, such as `TAR` or `_TAR` to the Change Table name for all Change Tables with that name. In this case, the resulting Change Table name will be `EMPLOYEESTAR` or `EMPLOYEES_TAR`.

To globally add a prefix or suffix:

1. Select **Add <Prefix/Suffix> <String> to matching Change Table names**.
2. Click the word **Prefix** or **Suffix** and select one of these two from the list.
3. Click **[string]** to activate the field.
4. Type the characters you want as the prefix or suffix. If you want to include an underscore or other legal character to separate the prefix/suffix from the original name, you must add it as part of the character string.
5. Click **Finish** to add the rule to the [Global Rules](#) list.

Remove a prefix or suffix

Use the **Remove a prefix or suffix** option to remove a string of characters from the beginning or end of a Change Table name for all Change Tables that fit the definition you created in the [Transformation scope \(page 122\)](#) step.

For example, you can use this option to remove the letters `_REV` from the Change Table name for all Change Tables with the name `EMPLOYEES`. In this case the Change Table name in the target will be `EMPLOYEES`.

To globally remove a prefix or suffix:

1. Select **Remove <Prefix/Suffix> <String> from matching Change Table names**.
2. Click the word **Prefix** or **Suffix** and select one of these two from the list.
3. Click **[string]** to activate the field.
4. Type the characters you want to remove. If you want to remove an underscore or other legal character from the original name, you must add it as part of the character string.
5. Click **Finish** to add the rule to the [Global Rules](#) list.

Replace a prefix or suffix with different characters

Use the **Replace a prefix or suffix** option to replace a string of characters with a different string of characters. You determine whether to replace the characters at the beginning or end of a Change Table name for all Change Tables that fit the definition you created in the [Transformation scope \(page 122\)](#) step.

For example, you can use this option to replace the letters `_ORIG` with `_REPL` in the Change Table names for all Change Tables called `EMPLOYEE_ORIG`. In this case the Change Table name in the target will be `EMPLOYEE_REPL`.

To globally replace a prefix or suffix:

1. Select **Replace <Prefix/Suffix> <String> by <String> for all matching Change Table names**.
2. Click the word **Prefix** or **Suffix** and select one of these two from the list.
3. Click the first **[string]** to activate the field.
4. Type the characters from the existing (source) schema that you want to replace. If you want to include an underscore or other legal character from the original name in the string that you want to replace, you must add it as part of the character string.
5. Click the second **[string]** to activate the field.
6. Type the characters you want to use in the target. These characters replace the original (source) characters in the target.
7. Click **Finish** to add the rule to the [Global Rules](#) list.

Convert Change Table name to uppercase

Use the convert to uppercase option to convert a Change Table name to all upper case. For example:

- Table_cat, becomes TABLE_CAT
- Change Table_cat, becomes TABLE_CAT
- taBlE_Cat, becomes TABLE_CAT

To globally change the Change Table name to all uppercase:

1. Select **Convert Change Table name to uppercase**.
2. Click **Finish** to add the rule to the [Global Rules](#) list.

Convert Change Table name to lowercase

Use the convert to lowercase option to convert a Change Table name to all lower case. For example:

- Table_cat, becomes Change Table_cat
- TABLE_CAT, becomes Change Table_cat
- taBlE_Cat, becomes Change Table_cat

To globally change the Change Table name to all lowercase:

1. Select **Convert Change Table name to lowercase**.
2. Click **Finish** to add the rule to the [Global Rules](#) list.

Rename Change Table (expression)

Use the **Rename Change Table to [expression]** option to change the name of all Change Tables that fit the definition you created in the [Transformation scope \(page 122\)](#) step. For example, if you have a Change Table called EMPLOYEE and want to change all instances of this name as defined in the previous step it to EMP.

To change the Change Table name:

1. Select **Rename Change Table to: [expression]**
2. Click the button to the right of the **Rename Change Table** option to open the Expression Editor. For information on how to use the Expression Editor, see [Using the Expression Builder \(page 149\)](#). Then go to step 4.
or
Click **[expression]** to activate the field and continue with step 3.
3. Type an SQLite expression or a string (in quotes) to rename the table. For example:
 - "New_Change_Table_Name"
 - 'PREF_'||\$AR_M_SOURCE_TABLE_NAME||'_SUFF'

You can use the following metadata in the SQLite expression:

- \$AR_M_SOURCE_COLUMN_DATATYPE
- \$AR_M_SOURCE_COLUMN_NAME
- \$AR_M_SOURCE_SCHEMA
- \$AR_M_SOURCE_TABLE_NAME

Rename Change Table schema

This transformation is only available when the Store Changes replication option is turned on.

If your transformation type is **Rename Change Table schema**, you can do the following:

- [Rename Change Table schema to \(string\) \(page 141\)](#)
- [Add a prefix or suffix \(page 141\)](#)
- [Remove a prefix or suffix \(page 142\)](#)
- [Replace a prefix or suffix with different characters \(page 142\)](#)
- [Convert Change Table schema name to uppercase \(page 143\)](#)
- [Convert Change Table schema name to lowercase \(page 143\)](#)
- [Rename Change Table schema \(expression\) \(page 143\)](#)

Rename Change Table schema to (string)

Use the **Rename Change Table schema to: [string]** option to change the name of all Change Table schemas that you defined in the [Transformation scope \(page 122\)](#) step to a different name. For example, if you have a Change Table schema called `EMPLOYEE` and want to change all instances of this name to `EMP` then enter the string `EMP`. You can enter any string in this field.

Add a prefix or suffix

Use the **Add a prefix or suffix** option to add additional characters to the beginning or end of the Change Table schema name for all tables that fit the definition you created in the [Transformation scope \(page 122\)](#) step. For example, if the Change Table schema name is `EMPLOYEES`, you can add a suffix, such as `TAR` or `_TAR` to the Change Table schema name for all Change Table schemas with that Change Table schema name. In this case, the resulting Change Table schema name will be `EMPLOYEESTAR` or `EMPLOYEES_TAR`.

To globally add a prefix or suffix:

1. Select **Add <Prefix/Suffix> Insert Characters to matching Change Table schema names**.
2. Click the word **Prefix** or **Suffix** and select one of these two from the list.
3. Click **[string]** to activate the field.
4. Type the characters you want as the prefix or suffix. If you want to include an underscore or other legal character to separate the prefix/suffix from the original name, you must add it as part of the character string.
5. Click **Finish** to add the rule to the [Global Rules](#) list.

Remove a prefix or suffix

Use the **Remove a prefix or suffix** option to remove a string of characters from the beginning or end of a Change Table schema name for all tables that fit the definition you created in the [Transformation scope \(page 122\)](#) step.

For example, you can use this option to remove the letters `_REV` from the Change Table schema name for all Change Table schemas with the name `EMPLOYEES`. In this case the Change Table schema name in the target will be `EMPLOYEES`.

To globally remove a prefix or suffix:

1. Select **Remove <Prefix/Suffix> <String> from matching Change Table schema names**.
2. Click the word **Prefix** or **Suffix** and select one of these two from the list.
3. Click **[string]** to activate the field.
4. Type the characters you want to remove. If you want to remove an underscore or other legal character from the original name, you must add it as part of the character string.
5. Click **Finish** to add the rule to the [Global Rules](#) list.

Replace a prefix or suffix with different characters

Use the **Replace a prefix or suffix** option to replace a string of characters with a different string of characters. You determine whether to replace the characters at the beginning or end of a Change Table schema name for all tables that fit the definition you created in the [Transformation scope \(page 122\)](#) step.

For example, you can use this option to replace the letters `_ORIG` with `_REPL` in the Change Table schema names for all Change Table schemas called `EMPLOYEE_ORIG`. In this case the Change Table schema name in the target will be `EMPLOYEE_REPL`.

To globally replace a prefix or suffix:

1. Select **Replace <Prefix/Suffix> <String> by <String> for all matching schema names**.
2. Click the word **Prefix** or **Suffix** and select one of these two from the list.
3. Click the first **[string]** to activate the field.
4. Type the characters from the existing (source) Change Table schema name that you want to replace. If you want to include an underscore or other legal character from the original name in the string that you want to replace, you must add it as part of the character string.
5. Click the second **[string]** to activate the field.

6. Type the characters you want to use in the target. These characters replace the original (source) characters in the target.
7. Click **Finish** to add the rule to the [Global Rules](#) list.

Convert Change Table schema name to uppercase

Use the convert to uppercase option to convert a Change Table schema name to all upper case. For example:

- Table_cat, becomes TABLE_CAT
- table_cat, becomes TABLE_CAT
- taBlE_Cat, becomes TABLE_CAT

To globally change the Change Table schema name to all uppercase:

1. Select **Convert Change Table schema name to uppercase**.
2. Click **Finish** to add the rule to the [Global Rules](#) list.

Convert Change Table schema name to lowercase

Use the convert to lowercase option to convert a Change Table schema name to all lower case. For example:

- Table_cat, becomes table_cat
- TABLE_CAT, becomes table_cat
- taBlE_Cat, becomes table_cat

To globally change the Change Table schema name to all lowercase:

1. Select **Convert Change Table schema name to lowercase**.
2. Click **Finish** to add the rule to the [Global Rules](#) list.

Rename Change Table schema (expression)

Use the **Rename Change Table schema to [expression]** option to change the name of all tables that fall within the scope you defined in the [Transformation scope \(page 122\)](#) step. For example, if you have a Change Table schema called EMPLOYEE and want to change all instances of the Change Table schema name as defined in the previous step to EMP.

To rename the Change Table schema:

1. Select **Rename Change Table schema to**.
2. Click the button to the right of the **Rename Change Table schema to** option to open the Expression Editor. For information on how to use the Expression Editor, see [Using the Expression Builder \(page 149\)](#).
or
Click **[expression]** to activate the field and continue with step 3.
3. Type an SQLite expression or a string (in quotes) to rename the table. For example:

- "New_Change_Table_Schema"
- 'PREF_'||\$AR_M_SOURCE_SCHEMA||'_SUFF'

You can use the following metadata in the SQLite expression:

- \$AR_M_SOURCE_COLUMN_DATATYPE
 - \$AR_M_SOURCE_COLUMN_NAME
 - \$AR_M_SOURCE_SCHEMA
 - \$AR_M_SOURCE_TABLE_NAME
4. When you're done, click **Next** to see a summary of your rule and replace the default name and description, or **Finish** to add the rule to the [Global Rules](#) list.

Starting the Global Filter Rules wizard

You define global filters using the Global Filter Rules wizard.

To start the Global Filters wizard:

1. Open the task for which you want to create a global transformations or a global filter. You can click **View Task** above the Tasks list or double-click the task.
2. If you are not in the Designer mode, click **Designer** at the top right of the screen. For more information on the Designer mode, see [Designer mode \(page 225\)](#).
3. In Designer mode, click **Global Rules**. The [Global Rules](#) window opens.
4. Click the **New Rule** toolbar button and select **Filter**. The **New Filter Rule** wizard opens.
5. Enter the information to define a global transformation rule. The first step is selecting the [Filter type \(page 144\)](#).

Filter type

In the **Filter Type** screen of the **New Filter Rule** wizard, you define the type of filtering you want to be performed.

- **Filter by columns** - Only include records that match specific column data. For example, only include records where Birth Date is later than 02-Feb-2021.



Filtering by columns containing changeable values (e.g. Age) may result in inconsistent data on the target.

- **Filter by record attributes** - Only include records that match specific record attributes. For example, only include UPDATE or INSERTED records.

Filter scope

In the **Filter Scope** screen of the **New Filter Rule** wizard, you define which tables will be filtered. For example, you can limit the filter to all tables that contain the word SALARY as part of their name.



The options displayed in this screen depend on selected [filter type](#).

Scope options when filtering by columns

The following table describes the options that are available when filtering by columns.

Columns filtering scope

Option	Description
Schema name is like %	<p>Leave the % sign to apply the rule to all source schemas (the default).</p> <p>Alternatively, enter a custom string. The rule will only be applied to columns in schemas that match the specified string.</p> <p>For example, enter <code>HR</code> to include only columns that belong to schema <code>HR</code>.</p> <p>You can use the % symbol as a wildcard. For example, specifying <code>H%</code> will include all columns in tables that belong to schemas beginning with the letter <code>H</code>, such as <code>HR</code>, <code>HELLO</code>, or <code>HQ</code>.</p> <p>The % wildcard can be used in any position. For example, if you use it at the beginning (<code>%H</code>), then the rule will be applied to all columns in schemas that end with the letter <code>"H"</code>. The % symbol can also be used in the middle of a string.</p> <div data-bbox="405 1077 474 1146" data-label="Image"></div> <p><i>If you are using an Oracle target, you must enter a schema that exists on the target endpoint. Qlik Replicate does not create new schemas on an Oracle endpoint. If you want to use a new schema for the target, create the schema on the Oracle endpoint before running the task. For more information, see the topic "Configuring an Oracle database as a Qlik Replicate Target" in the Qlik Replicate User and Reference Guide.</i></p>
Table name is like %	<p>Leave the % sign to apply the rule to all source tables (the default).</p> <p>Alternatively, enter a custom string. The rule will only be applied to columns in tables that match the specified string.</p> <p>You can use the % symbol as a wildcard. For example, specifying <code>J%</code> will include all columns in tables with names beginning with the letter <code>J</code>, such as <code>JOBS</code>, <code>JOBS_HISTORY</code>, or <code>JACKSONVILLE</code>.</p> <p>The % wildcard can be used in any position. For example, if you use it at the beginning (<code>%H</code>), then the rule will be applied to all columns in tables that end with the letter <code>"H"</code>. The % symbol can also be used in the middle of a string.</p>

Option	Description
Column name is like %	<p>Leave the % sign to apply the rule to all source columns (the default).</p> <p>Alternatively, enter a custom string. The rule will only be applied to columns that match the specified string.</p> <p>You can use the % symbol as a wildcard. For example, specifying N% will include all columns with names beginning with the letter N, such as NAME, NAME_FIRST, or NAME_LAST.</p> <p>The % wildcard can be used in any position. For example, if you use it at the beginning (%n), then the rule will be applied to all columns that end with the letter "n". The % symbol can also be used in the middle of a string.</p>
Data type is	<p>Optionally, select a specific data type from the drop-down list. Make sure the data type you select is compatible with the data in that column.</p> <p>For a description of Qlik Replicate data types, information about data type mapping from the native endpoint to Qlik Replicate, or for a list of endpoints supported by Qlik Replicate, see the Qlik Replicate User and Reference Guide.</p>
Scope expression	<p>Click Advanced options to define a scope expression using the expression builder.</p>

Scope options when filtering by record attributes

The following table describes the options that are available when filtering by record attributes.

Record attributes filtering scope

Option	Description
Schema name is like %	<p>Leave the % sign to apply the rule to all source schemas (the default).</p> <p>Alternatively, enter a custom string. The rule will only be applied to columns in schemas that match the specified string.</p> <p>For example, enter HR to include only columns that belong to schema HR.</p> <p>You can use the % symbol as a wildcard. For example, specifying H% will include all columns in tables that belong to schemas beginning with the letter H, such as HR, HELLO, or HQ.</p> <p>The % wildcard can be used in any position. For example, if you use it at the beginning (%H), then the rule will be applied to all columns in schemas that end with the letter "H". The % symbol can also be used in the middle of a string.</p> <div style="border: 1px solid #ccc; padding: 10px; margin-top: 10px;"> <p> <i>If you are using an Oracle target, you must enter a schema that exists on the target endpoint. Qlik Replicate does not create new schemas on an Oracle endpoint. If you want to use a new schema for the target, create the schema on the Oracle endpoint before running the task. For more information, see the topic "Configuring an Oracle database as a Qlik Replicate Target" in the Qlik Replicate User and Reference Guide.</i></p> </div>
Table name is like %	<p>Leave the % sign to apply the rule to all source tables (the default).</p> <p>Alternatively, enter a custom string. The rule will only be applied to columns in tables that match the specified string.</p> <p>You can use the % symbol as a wildcard. For example, specifying J% will include all columns in tables with names beginning with the letter J, such as JOBS, JOBS_HISTORY, or JACKSONVILLE.</p> <p>The % wildcard can be used in any position. For example, if you use it at the beginning (%H), then the rule will be applied to all columns in tables that end with the letter "H". The % symbol can also be used in the middle of a string.</p>
Scope expression	Click Advanced options to define a scope expression using the expression builder.

After defining the filter scope, click **Next** to proceed to the [Transformation action \(page 126\)](#) screen.

Filter action

In the **Filter action** screen, you create an expression for selecting records based on their column data (when defining a **Filter by columns**) or based on their attributes (when defining a **Filter by record attributes**). Only records that fall within the [Filter scope \(page 144\)](#) will be affected.

Filter action for columns

Use the Expression Builder to define a record selection expression based on column data. Click the **fx** button to open the Expression Builder.

The Expression Builder opens displaying the **Data** tab. Define an expression using the [available parameters](#).

Example (Assumes that "Age" is the column being filtered):

Create the following expression to select all records for citizens between the age of 50 and 80.

```
$AR_M_SOURCE_COLUMN_DATA > '60' AND $AR_M_SOURCE_COLUMN_DATA < '70'
```

Filter action for record attributes

Use the Expression Builder to define a record selection expression based on record attributes. Click the **fx** button to open the Expression Builder.

The Expression Builder opens displaying the **Headers** tab. Define an expression using the [available Header columns](#).



Header columns are not relevant for Full Load tasks. They are only relevant when the task's Apply Changes or Store Changes replication options are enabled.

Example:

Create the following expression to select UPDATED records only:

```
$AR_H_OPERATION='UPDATE'
```

See also: [Using the Expression Builder \(page 149\)](#)

Managing global rules

The **Global Rules** window lists the names and descriptions of all rules that are defined for the Qlik Replicate instance you are working with. This is where you go to edit, delete, reposition, or activate/deactivate rules.

Editing global rules



You cannot change the name of a transformation rule

To edit a global rule:

1. In the **Global Rules** window, select the rule you want to edit.
2. Click the **Edit** button or double-click the rule.
The **Edit Existing Transformation Rule** wizard or **Edit Existing Filter Rule** wizard opens.

3. Make any changes you need in the wizard.

For information on how to configure a global transformation, see the following topics:

- [Transformation type \(page 120\)](#)
- [Transformation scope \(page 122\)](#)
- [Transformation action \(page 126\)](#)

For information on how to configure a global filter, see the following topics:

- [Filter type \(page 144\)](#)
- [Filter scope \(page 144\)](#)
- [Filter action \(page 147\)](#)

Deleting global rules

To delete a global rule:

1. In the **Global Rules** window, select the rule you want to delete.
2. Click the **Delete** button.
3. When prompted for confirmation, click **OK**.
The rule is removed from the list and deleted from the system.

Positioning global rules

Rules are applied in the order they appear. To prevent conflicts therefore, ensure that rules are arranged in a logical order. You can use the up/down arrows to rearrange the rule order.

Activating or deactivating global rules

To activate or deactivate a rule, select or clear the check box in the **Activate** column as needed.

9.3 Using the Expression Builder

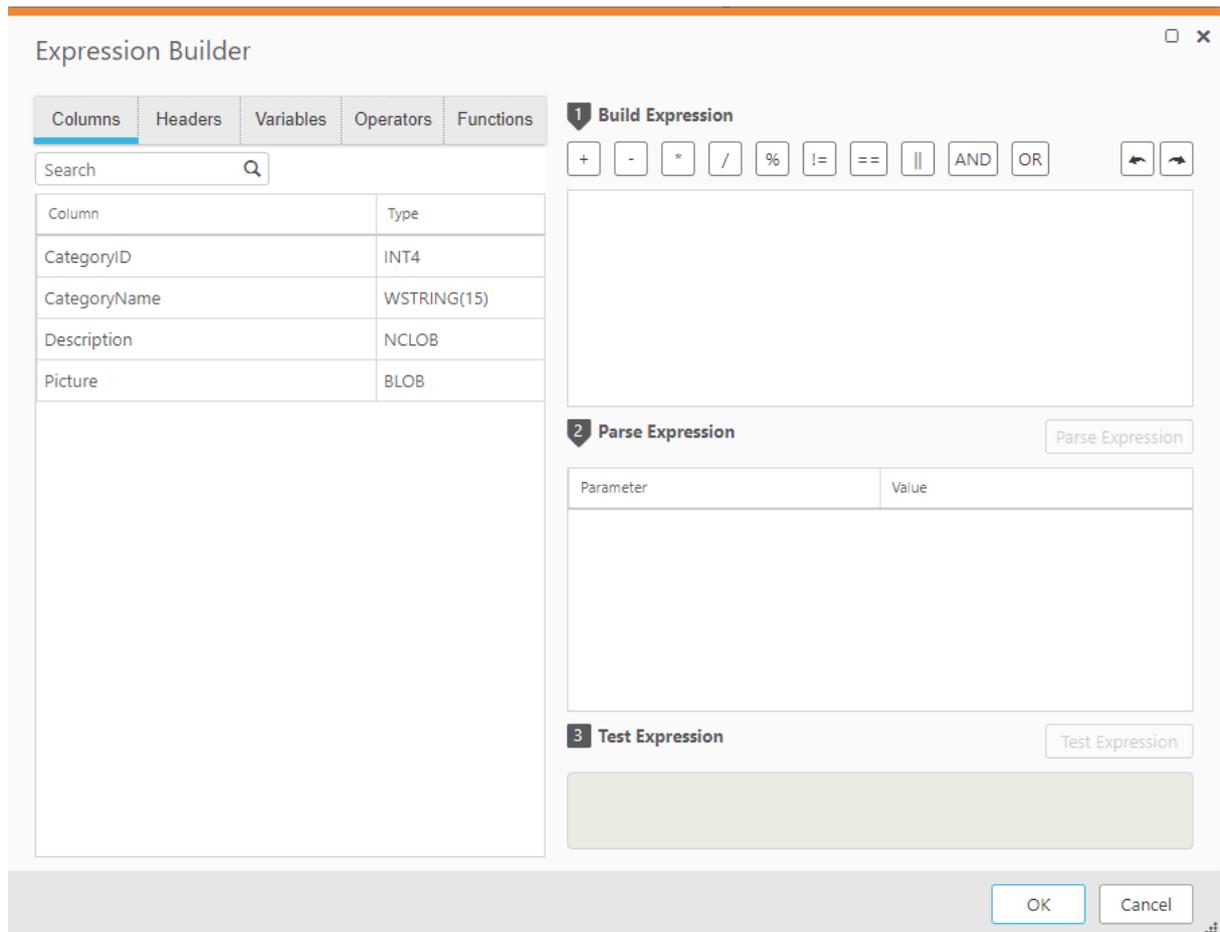
The Qlik Replicate Expression Builder provides an easy way to build an expression. It provides you with easy access to the required elements for your expression without having to type out any information manually. You can use the Expression Builder to create advanced expressions for [filters](#) and transformations such as [renaming a schema](#), [table](#), or [column](#).

The following topics describe the Expression Builder:

- [Overview of the Expression Builder \(page 149\)](#)
- [Build an expression \(page 151\)](#)
- [Parse an expression \(page 152\)](#)
- [Test an expression \(page 152\)](#)
- [Using elements in the Expression Builder \(page 154\)](#)

Overview of the Expression Builder

The following is an example of the Expression Builder with its four main parts shown. The Expression Builder you are working with may look different depending on whether you want to build an expression for a filter, a transformation, or a global transformation.



Expression Builder for filters, transformations, and global transformations

The following sections describe the tasks you can perform in each part of the Expression Builder:

- **Elements Pane (on the left):** This pane contains elements that you can add to an expression. Select elements and move them into the Expression Builder box to create the expression. For more information, see [Build an expression \(page 151\)](#).

The Elements Pane contains the following tabs:

- **Metadata** (available only when working with Global transformations)
- **Input** (available only when working with transformations or filters)
- **Header** (for Global transformations, this tab is available only when you select **Add Column**)
- **Variables**
- **Operators**
- **Functions**

For more information on these elements, see [Using elements in the Expression Builder \(page 154\)](#).

- **Build Expression Panel:** The Build Expression Panel is where you put together the expression you are building. You move elements, such as columns or operators into the box. You can also type all or part of an expression in this box.

For more information, see [Build an expression \(page 151\)](#).

- **Parse Expression Panel:** This panel displays the parameters for the expression. After you build the expression, click **Parse Expression** to list the expression parameters. You can then enter a value or argument for each of the parameters. For more information, see [Parse an expression \(page 152\)](#). The top part of the Expression panel contains the **Operator** toolbar. This toolbar contains the most common operators. Click the operator you want to use to add it to the expression. You can also add operators from the Element Pane, **Operators** tab.
- **Test Expression Panel:** This panel displays the results of a test that you can run after you provide values to each of the parameters in your expression. For more information, see [Test an expression \(page 152\)](#).

Build an expression

The first step in using the expression builder is to build an expression. The expression that you build is displayed in the top section of the right pane. You can open the Expression when:

- You define [Defining transformations for a single table/view \(page 93\)](#) for a single table.
- You define [Filters](#) for a single table.
- You use the Global transformations dialog box to [Rename Schema \(page 127\)](#), [Rename Table \(page 130\)](#), [Rename Column \(page 133\)](#), or [Add column \(page 136\)](#).

Note: To add operators to your expression, you can use the **Operator** tab in the Element pane or the Operator buttons at the top of the Build Expression panel or any combination of these. See [Operators \(page 156\)](#) and [Operator toolbar \(page 151\)](#).

For example, to create an expression that will combine the first name and last name, do the following:

1. In the **Input Columns** tab add the FIRST_NAME column to the **Build Expression** box.
2. Click the concatenate (||) operator from the **Operator** bar at the top of the Build Expression box.
3. In the Input Columns tab add the LAST_NAME column into the **Build Expression** box.

To build an expression:

1. In the Elements Pane, select any element you want to include in your expression. For information on the elements you can use in an expression, see [Functions \(page 160\)](#).
2. Add an element to the **Build Expression** panel by selecting it and then clicking the arrow to the right of the element.
3. Continue to add elements as needed.

Operator toolbar

The Operator toolbar is above the Build Expression box. It contains the most common operators so you can easily add them to an expression.

The following operators are available in the Operator toolbar: addition, subtraction, multiplication, division, percentage, not equal, equal, concatenate, AND, OR.



For information on these operators, see [Operators \(page 156\)](#).

To use the Operator toolbar:

1. Click the place in the Build Expression box where you want to add the operator.
2. Click the operator you want to add. It is added to the expression.

Parse an expression

You can parse an expression to determine its parameters and to determine whether the expression is valid.

To parse an expression:

1. In the Expression Builder window, create an expression as described in [Build an expression \(page 151\)](#).
2. Click **Parse Expression**.
If the expression is not valid, an error message is written in red at the bottom of the Expression Builder window.
If the expression is valid, the expression parameters are displayed in the **Parameter** column in the **Parse Expression** section. See the figure under [Test an expression \(page 152\)](#).
3. Type a valid value for each of the parameters in the **Value** column to [Test an expression \(page 152\)](#).
For example, type `John` for the `FIRST_NAME` and `Smith` for the `LAST_NAME` in the **Value** column.
Once you type in values, you can [Test an expression \(page 152\)](#).

Test an expression

You can use the Qlik Replicate Test procedure to display the results of a test expression. The following figure is an example of a built expression that is evaluated and contains a test result.

1 Build Expression

+ - * / % != == || AND OR

`$First_Name || $Last_Name`

2 Evaluate Expression

Evaluate

Parameter	Value
<code>\$First_Name</code>	<input type="text" value="Mike"/>
<code>\$Last_Name</code>	<input type="text" value="Smith"/>

3 Test Expression

Test

MikeSmith

Test Expression dialog

To test an expression:

1. From the Expression Builder window, [Build an expression \(page 151\)](#).
2. Click **Evaluate**. See [Parse an expression \(page 152\)](#) for more information.

3. View the parameters that are displayed. If your expression is not valid, an error message is displayed. See [Parse an expression \(page 152\)](#).
4. Type values for each parameter then click **Test** to see the calculated expression. For example, type John for FIRST_NAME and Smith for LAST_NAME. The result displayed is JohnSmith. If you want a space between the words add it to the end of the FIRST_NAME value or the beginning of the LAST_NAME value.
Note: Testing calls to the source_lookup and target_lookup functions is not supported.

Using elements in the Expression Builder

You can use the following types of elements to build expressions for transformations, filters, and global transformations. Select the appropriate tab to select the elements.

- [Columns \(transformations and filters only\) \(page 154\)](#)
- [Data \(global transformations only\) \(page 154\)](#)
- [Metadata \(Global transformations only\) \(page 155\)](#)
- [Variables \(page 155\)](#)
- [Operators \(page 156\)](#)
- [Functions \(page 160\)](#)
- [Headers \(page 173\)](#)
- [User-defined transformations \(page 177\)](#)

Data (global transformations only)

In this tab, you can create an expression that leverages modified source column data and/or the original source column data. Modified column data is data that has been replaced or modified using a global transformation (e.g. **Replace column value**) whereas the original source column data is the data as it appears in the source database.

The following options are available:

- `$AR_M_MODIFIED_COLUMN_DATA` - Use to specify column data that has been replaced or modified using a global transformation.
- `$AR_M_SOURCE_COLUMN_DATA` - Use to specify the original column data.

Example:

For a source column called "Age", the following expression can be defined to exclude all records for citizens between the age of 50 and 80.

```
$AR_M_SOURCE_COLUMN_DATA > '50' AND $AR_M_SOURCE_COLUMN_DATA < '80'
```

Columns (transformations and filters only)

This tab lists the columns for the table you are working with. The table you are working with is the table you selected when you opened the **Table Settings** dialog box.



Expressions containing a condition for a `NUMBER` exceeding 19 digits are not valid. For example, the following expression is not valid:

```
case when $ID==12345678910111213149 then '1' else '0' end
```

Metadata (Global transformations only)

The **Metadata** tab contains the following variables that you can use in an expression. Variables with `MODIFIED` in their name can be used to specify metadata that has been replaced or modified using a global transformation (such as **Convert data type**) whereas variables names *without* `MODIFIED` can be used to specify the original metadata as it is defined in the source database.

- `AR_M_MODIFIED_SCHEMA` - The modified source schema name.
- `AR_M_MODIFIED_TABLE_NAME` - The modified source table name.
- `AR_M_MODIFIED_COLUMN_NAME` - The modified column name in the source table.
- `AR_M_MODIFIED_COLUMN_SUBTYPE_NAME` - The modified data type subtype of a column in the source table.
- `AR_M_MODIFIED_DATATYPE_NAME` - The modified data type of a column in the source table.
- `AR_M_MODIFIED_DATATYPE_LENGTH` - The modified data type length of a column in the source table.
- `AR_M_MODIFIED_DATATYPE_PRECISION` - The modified data type precision of a column in the source table.
- `AR_M_MODIFIED_DATATYPE_SCALE` - The modified data type scale of a column in the source table.
- `AR_M_SOURCE_SCHEMA` - The name of the source schema.
- `AR_M_SOURCE_TABLE_NAME` - The name of the source table.
- `AR_M_SOURCE_COLUMN_NAME` - The name of a column in the source table.
- `AR_M_SOURCE_DATATYPE_NAME` - The data type of a column in the source table.
- `AR_M_SOURCE_DATATYPE_LENGTH` - The data type length of a column in the source table.
- `AR_M_SOURCE_DATATYPE_PRECISION` - The data type precision of a column in the source table.
- `AR_M_SOURCE_DATATYPE_SCALE` - The data type scale of a column in the source table.

Example: Using metadata variables in the Rename Column transformation

To rename all columns named "product" to "source_schema.table_name", enter "product" in the **Column name is like** field (in the [Transformation scope \(page 122\)](#) screen) and then enter the following expression in the **Rename column to** field (in the [Transformation action \(page 126\)](#) screen):

```
$AR_M_SOURCE_SCHEMA || "." || $AR_M_SOURCE_TABLE_NAME
```

Variables

Your expression can contain any of the variables (which will be replaced during runtime) described in the table below.

General variables

General variables

Variable Name	Description	Data Type
AR_V_HOST_NAME	The host name of the machine on which Qlik Replicate Server is installed.	STRING (50)
AR_V_SOURCE_NAME	The logical name of the source endpoint defined in the endpoint settings.	STRING (50)
AR_V_ERROR_TEXT	The error message returned by the task	STRING
AR_V_TARGET_NAME	The logical name of the target endpoint defined in the endpoint settings.	STRING (50)
AR_V_TASK_NAME	The task name.	STRING (50)
AR_V_TASK_UUID	A unique string (Universal Unique Identifier) that identifies the task.	STRING (50)
AR_V_RELOAD_TIME	The time the source tables were reloaded.	DATETIME (6)
AR_V_START_TIME	The time the task started.	DATETIME (6)

Operators

The sections below describe the SQLite operators you can use to build an expression with the Expression builder. The Expression builder divides the operators into the following categories:

- [Strings \(page 156\)](#)
- [Logical \(page 157\)](#)
- [Mathematical \(page 159\)](#)



With the exception of table-level transformations, all operator symbols must be preceded by a space and followed by a space. For example, the expression for concatenating a first and last name should be specified like this:

FIRST_NAME || LAST_NAME

And not like this:

FIRST_NAME|LAST_NAME

Strings

You can use the following string:

||

Name: Concatenate strings.

Examples:

FIRST_NAME || LAST_NAME

PHONE_NUMBER || '<office only>' (adds the string office only to the telephone number).

Logical

The following table describes the logical SQLite operators used by the Qlik Replicate Expression Builder.

Logical operators

Operator	Description
!= or <>	Is not equal to \$SALARY!=100000
IS	Is the same as \$HIRE_DATE IS 2014-09-29 IS functions the same as = unless one or both of the operands are NULL. In this case, if both operands are NULL, then the IS operator evaluates to 1 (true). If one operand is NULL and the other is not, then the IS operator evaluates to 0 (false).
IS NOT	Is not the same as \$HIRE_DATE IS NOT 2014-09-29 IS NOT functions the same as != unless one or both of the operands are NULL. In this case, if both operands are NULL, the IS NOT operator evaluates to 0 (false). If one operand is NULL and the other is not, then the IS NOT operator evaluates to 1 (true).
IN	The IN operator takes a single scalar operand on the left and a vector operand on the right formed by an explicit list of zero or more scalars or by a single subquery. When the right operand of an IN operator is a subquery, the subquery must have a single result column. When the right operand is an empty set, the result of IN is false regardless of the left operand and even if the left operand is NULL. SQLite allows the parenthesized list of scalar values on the right-hand side of an IN operator to be an empty list but most other SQL endpoint engines and the SQL92 standard require the list to contain at least one element.

Operator	Description
LIKE	<p>The LIKE operator does a pattern matching comparison. The operand to the right of the LIKE operator contains the pattern and the left operand contains the string to match against the pattern. A percent symbol ("%") in the LIKE pattern matches any sequence of zero or more characters in the string. An underscore ("_") in the LIKE pattern matches any single character in the string. Any other character matches itself or its lower/upper case equivalent. (By default SQLite only understands upper/lower case for ASCII characters. The LIKE operator is case sensitive by default for unicode characters that are beyond the ASCII range.</p> <p>For example, the expression 'a' LIKE 'A' is TRUE but 'æ' LIKE 'Æ' is FALSE.)</p> <p>LIKE can be preceded by the NOT keyword.</p>
CASE	<p>Evaluates a list of conditions and returns one of multiple possible result expressions.</p> <p>Example 1:</p> <pre>WHEN \$NEWEST = 'Y' THEN '1' ELSE '0' END</pre> <p>Example 2:</p> <pre>case length(\$month) when 2 then \$year \$month when 1 then \$year 0 \$month end</pre>
GLOB	<p>The GLOB operator acts in the same way as the LIKE operator but uses the UNIX file globbing syntax for its wildcards. GLOB is case sensitive.</p> <p>GLOB can be preceded by the NOT keyword to invert the sense of the test. The infix GLOB operator is implemented by calling the function glob(Y,X) and can be modified by overriding that function.</p>
MATCH	<p>The MATCH operator is a special syntax for the match() application-defined function. The default match() function implementation raises an exception and is not really useful for anything. But extensions can override the match() function with more helpful logic.</p>
REGEXP	<p>The REGEXP operator is a special syntax for the regexp() user function. No regexp() user function is defined by default and so use of the REGEXP operator will normally result in an error message.</p>
AND	<p>Both operands are true.</p> <pre>\$MANAGER_ID AND EMPLOYEE ID >100</pre>
OR	<p>Either operand is true.</p> <pre>\$MANAGER_ID OR EMPLOYEE ID >100</pre>

Operator	Description
<<	Bitwise shift left. x << n A bitwise shift to the left of x by n bits.
>>	Bitwise shift right. x >> n A bitwise shift to the right of x by n bits.
&	Unary and
	Unary or
<	Is less than. \$SALARY<100000
<=	Is less than or equal to \$SALARY<=100000
>	Is greater than \$SALARY>100000
>=	Is more than or equal to \$SALARY>=100000
= or ==	Is equal to \$SALARY=100000

Mathematical

The following table describes the mathematical SQLite operators used by the Expression Builder.

Mathematical operators

Operator	Description
+	Adds two values together. DEPARTMENT_ID+100 (adds 100 to each ID number). Any column used in an expression with this operator must be a numeric data type.
-	Subtracts a value from another value. MANAGER_ID-100 (subtracts 100 from each ID number). Any column used in an expression with this operator must be a numeric data type.

Operator	Description
%	Uses the remainder of a division expression as the value. %SALARY/7 (Divides the value of the Salary column by 7 and uses any remainder from the expression as the column value).
/	Divides one value into another. SALARY/.16 (Divides the value of the Salary column by .16. Note: If the two values in the division expression are integers (two NUMERIC columns with no digits after the decimal) and the result is a fractional value, the result returned will be 0.
*	SALARY*.16 (Multiplies the value of the Salary column by .16. This could be used to calculate taxes that are subtracted from a salary).

Functions

The sections below describe the SQLite functions you can use to build an expression with the Expression builder. The Expression builder divides the functions into the following categories:

- [Strings \(page 160\)](#)
- [LOBs \(page 161\)](#)
- [Numeric \(page 161\)](#)
- [NULL check \(page 162\)](#)
- [Date and Time \(page 163\)](#)
- [Data Enrichment \(page 165\)](#)
- [Operation \(page 170\)](#)
- [Other Functions \(page 171\)](#)
- [Hash \(page 172\)](#)
- [User-defined transformations \(page 177\)](#)

Strings

The following table describes the string functions used by the Expression Builder in Enterprise Manager.

String functions

Function	Description
lower(x)	The lower(x) function returns a copy of string x with all characters converted to lower case. The default built-in lower() function works for ASCII characters only.
ltrim(x,y)	The ltrim(x,y) function returns a string formed by removing all characters that appear in y from the left side of x. If there is no value for y, ltrim(x) removes spaces from the left side of x.
replace(x,y,z)	The replace(x,y,z) function returns a string formed by substituting string z for every occurrence of string y in string x.

Function	Description
<code>rtrim(x,y)</code>	The <code>rtrim(x,y)</code> function returns a string formed by removing all characters that appear in <code>y</code> from the right side of <code>x</code> . If there is no value for <code>y</code> , <code>rtrim(x)</code> removes spaces from the right side of <code>x</code> .
<code>substr(x,y,z)</code>	The <code>substr(x,y,z)</code> function returns a substring of input string <code>x</code> that begins with the <code>y</code> -th character and which is <code>z</code> characters long. If <code>z</code> is omitted then <code>substr(x,y)</code> returns all characters through the end of the string <code>x</code> beginning with the <code>y</code> -th. The left-most character of <code>x</code> is number 1. If <code>y</code> is negative then the first character of the substring is found by counting from the right rather than the left. If <code>z</code> is negative then the <code>abs(z)</code> characters preceding the <code>y</code> -th character are returned. If <code>x</code> is a string then characters indices refer to actual UTF-8 characters. If <code>x</code> is a BLOB then the indices refer to bytes.
<code>trim(x,y)</code>	The <code>trim(x,y)</code> function returns a string formed by removing all characters that appear in <code>y</code> from both sides of <code>x</code> . If there is no value for <code>y</code> , <code>trim(x)</code> removes spaces from both sides of <code>x</code> .
<code>replaceChars (X,Y,Z)</code>	<p>The <code>replaceChars (X, Y, Z)</code> function replaces any character in string <code>X</code> that also exists in string <code>Y</code> (characters to be replaced) with <code>Z</code> (replacement characters) in the same position. This is especially useful for removing non-valid characters from paths and file names.</p> <ul style="list-style-type: none"> • If string <code>Z</code> (replacement characters) does not include a character that has corresponding position in string <code>X</code>, it will be replaced with the first character in string <code>Z</code>. • If string <code>X</code> includes a character that does not exist in string <code>Z</code>, the original character will be left unchanged. <p>So, for example, specifying <code>replaceChars ("abcde", "abcd", "123")</code> would return <code>1231e</code>.</p>

LOBs

The following table describes the LOB functions used by the Expression Builder in Enterprise Manager.

LOB functions

Function	Description
<code>hex(x)</code>	The <code>hex()</code> function receives an argument as a BLOB and returns an upper-case hexadecimal string version of the BLOB content.
<code>randblob (N)</code>	The <code>randblob(N)</code> function returns an N-byte BLOB that contains pseudo-random bytes. If <code>N</code> is less than 1 then a 1-byte random BLOB is returned.
<code>zeroblob(N)</code>	The <code>zeroblob(N)</code> function returns a BLOB that consists of N bytes of 0x00.

Numeric

The following table describes the numeric functions used by the Expression Builder in Enterprise Manager.

Numeric functions

Function	Description
abs(x)	The abs(x) function returns the absolute value of the numeric argument X. Abs(x) returns NULL if x is NULL. Abs(x) returns 0.0 if x is a string or BLOB that cannot be converted to a numeric value.
random()	The random() function returns a pseudo-random integer between -9223372036854775808 and +9223372036854775807.
round(x,y)	The round(x,y) function returns a floating-point value x rounded to y digits to the right of the decimal point. If there is no value for y, it is assumed to be 0.
max(x,y...)	The multi-argument max() function returns the argument with the maximum value, or returns NULL if any argument is NULL. The multi-argument max() function searches its arguments from left to right for an argument that defines a collating function and uses that collating function for all string comparisons. If none of the arguments to max() define a collating function, then the BINARY collating function is used. Note that max() is a simple function when it has two or more arguments but operates as an aggregate function if it has a single argument.
min(x,y...)	The multi-argument min() function returns the argument with the minimum value. The multi-argument min() function searches its arguments from left to right for an argument that defines a collating function and uses that collating function for all string comparisons. If none of the arguments to min() define a collating function, then the BINARY collating function is used. Note that min() is a simple function when it has two or more arguments but operates as an aggregate function if it has a single argument.

NULL check

The following table describes the NULL check functions used by the Expression Builder in Enterprise Manager.

NULL check functions

Function	Description
coalesce(x,y...)	The coalesce() function returns a copy of its first non-NULL argument, it returns NULL if all arguments are NULL. Coalesce() have at least two arguments.
ifnull(x,y)	The ifnull() function returns a copy of its first non-NULL argument, it returns NULL if both arguments are NULL. Ifnull() must have exactly two arguments. The ifnull() function is the same as coalesce() with two arguments.
nullif(x,y)	The nullif(x,y) function returns a copy of its first argument if the arguments are different and returns NULL if the arguments are the same. The nullif(x,y) function searches its arguments from left to right for an argument that defines a collating function and uses that collating function for all string comparisons. If neither argument to nullif() defines a collating function then the BINARY is used.

Date and Time

The following table describes the Date and Time functions used by the Expression Builder in Enterprise Manager.

Date and Time functions

Function	Description
<code>date(timestring, modifier, modifier,...)</code>	Returns the date in the format YYYY-MM-DD.
<code>time(timestring, modifier, modifier,...)</code>	Returns the time in the format HH:MM:SS.
<code>datetime(timestring, modifier, modifier,...)</code>	Returns the date and time in the format YYYY-MM-DD HH:MM:SS.
<code>julianday(timestring, modifier, modifier,...)</code>	The <code>julianday()</code> function returns the number of days since noon in Greenwich on November 24, 4714 B.C.
<code>unixepoch(time-value, modifier, modifier,...)</code>	The <code>unixepoch()</code> function returns a unix timestamp - the number of seconds since 1970-01-01 00:00:00 UTC. The <code>unixepoch()</code> always returns an integer, even if the input time-value has millisecond precision.

Function	Description
<code>strftime(format, timestring, modifier, modifier...)</code>	<p>The <code>strftime()</code> routine returns the date formatted according to the format string specified as the first argument. It supports the following variables:</p> <ul style="list-style-type: none"> <code>%d</code>: day of month <code>%H</code>: hour 00-24 <code>%f</code>: ** fractional seconds SS.SSS <code>%j</code>: day of year 001-366 <code>%J</code>: ** Julian day number <code>%m</code>: month 01-12 <code>%M</code>: minute 00-59 <code>%s</code>: seconds since 1970-01-01 <code>%S</code>: seconds 00-59 <code>%w</code>: day of week 0-6 sunday==0 <code>%W</code>: week of year 00-53 <code>%Y</code>: year 0000-9999 <code>%%</code>: %

Examples

The expression builder provides you with a variety of options to build your own expression. You can use the regular operators with a date as well:

```
$HIRE_DATE < '2022-02-28'
```

Note that the right operand of the operator is contained between single quotes to prevent it being treated as a mathematical expression (2022 minus 2 minus 28).

Advanced examples using SQLite functions, modifiers and other operators

```
$HIRE_DATE < date('2022-02-28')
```

```
$DATE_1 < '2022-03-01' OR $DATE_1 > '2022-08-01'
```

```
$DATE_2 >= date('2022-03-01') AND $DATE_2 <= date('2022-08-01')
```

```
$HIRE_DATE < date('now','localtime','-1 year')
```

```
$HIRE_DATE <= date('now','utc')
```

```
$HIRE_DATE <= date('2022-02-28','+1 months')
```

```
$DATE_1 <= date($DATE_2,'+1 months')
```

List of valid modifiers in SQLite

The 'now' time string is the current date/datetime. You can specify an explicit date '2022-02-28' or you can use the date from a table column.

You can also apply multiple modifiers such as `datetime('now', '-3 hours', '+1 months')`, for example.

Modifier	Example	Result
	<code>datetime('now')</code>	2020-04-26 00:53:53
NNN days	<code>date('now', '+3 days')</code>	2020-04-29
NNN hours	<code>datetime('now', '-3 hours')</code>	2020-04-26 03:53:53
NNN minutes	<code>datetime('now', '+3 minutes')</code>	2020-04-26 00:56:53
NNN.NNNN seconds	<code>datetime('now', '-30 seconds')</code>	2020-04-26 00:54:23
NNN months	<code>date('now', '+3 months')</code>	2020-07-26
NNN years	<code>date('now', '-3 years')</code>	2017-04-26
start of month	<code>date('now', 'start of month')</code>	2020-04-01
start of year	<code>date('now', 'start of year')</code>	2020-01-01
start of day	<code>datetime('now', 'start of day')</code>	2020-04-26 00:00:00
weekday N	<code>date('now', 'weekday 6')</code>	2020-05-02
unixepoch	<code>datetime('1588965525', 'unixepoch')</code>	2020-05-08 19:18:45
localtime	<code>datetime('now', 'localtime')</code>	2020-04-26 10:53:53
utc	<code>datetime('now', 'utc')</code>	2020-04-25 08:53:53

Data Enrichment

Data Enrichment functions allow the selected source tables to be augmented with data from other records located in either the source or target endpoints. Practical applications of data enrichment functions include code lookup or master record lookup (e.g. social security number lookup to find a person's name).

You can enrich the target tables with supplemental data retrieved from the source or target endpoint by defining a transformation on the table. For more information about defining transformations on a single table, see [Defining transformations for a single table/view \(page 93\)](#).

Supported endpoints

Data enrichment is supported with the following endpoints only:

- Oracle
- Microsoft SQL Server
- Microsoft SQL Server (MS-CDC)
- Microsoft Azure SQL Database
- Microsoft Azure SQL Managed Instance
- Microsoft Azure SQL (MS-CDC)
- Google Cloud SQL for SQL Server
- Amazon RDS for SQL Server
- ODBC
- ODBC with CDC
- Teradata Database
- PostgreSQL
- Google Cloud SQL for PostgreSQL
- Google Cloud AlloyDB for PostgreSQL
- Netezza Performance Server
- Google Cloud BigQuery
- Amazon Redshift
- MySQL
- Google Cloud SQL for MySQL
- Amazon RDS for MySQL
- Google Cloud SQL for MySQL
- SingleStore
- PostgreSQL
- Amazon RDS for PostgreSQL
- Google Cloud SQL for PostgreSQL
- Google Cloud AlloyDB for PostgreSQL
- AWS Aurora Cloud for PostgreSQL
- SAP HANA
- SAP Application
- SAP Application (DB)
- SAP Sybase ASE
- IBM Informix
- IBM DB2 for z/OS
- IBM DB2 for LUW
- IBM DB2 for iSeries

Data Enrichment functions

The table below describes the source and target lookup functions, which can be used both for table transformations and for global transformations. For a description of the parameters available for these functions, see Input Parameters below.

Data Enrichment functions

Function	Description
source_lookup (TTL, 'SCHM', 'TBL', 'EXP', 'COND', COND_PARAMS)	Use to retrieve additional data from the source endpoint.
target_lookup (TTL, 'SCHM', 'TBL', 'EXP', 'COND', COND_PARAMS)	Use to retrieve additional data from the target endpoint.

Input parameters

The possible input parameters for the lookup functions are described in the table below. For a usage example, see [Data Enrichment example \(page 169\)](#).

Data Enrichment input parameters

Function	Description
TTL	<p>TTL (Time to Live) is the amount of time the 'COND' return value will be cached. Caching the 'COND' return value improves performance by reducing the frequency that Enterprise Manager needs to access the source/target endpoint. As there is no default, you must specify a TTL value, which can be one of the following:</p> <p><SECONDS> - The time to cache the 'COND' return value in seconds. Specify a short caching time (e.g. 3) for data that is frequently updated or a long caching time for data that rarely changes.</p> <p>'NO_CACHING' - Specify 'NO_CACHING' if you do not want to cache the 'COND' return value. This is recommended for data that is constantly updated (e.g. share prices).</p> <p>'NO_EXPIRATION' - For data that is never updated (e.g. a street name), specify 'NO_EXPIRATION' to store the Functions (page 160) return value permanently in the cache.</p>
'SCHM'	The schema name.
'TBL'	The table on which to perform the lookup.

Function	Description
'EXP'	<p>The expression to retrieve data from the lookup table.</p> <p>Note: The expression syntax must be native to the endpoint it accesses.</p> <p>The result should be a single column. Possible expressions include: <code>col1</code>, <code>col1+5</code>, <code>max(col1)</code>.</p> <p>Note: Full LOB columns are not supported. For information on including Limited-size LOB columns in the replication, see the description of the Metadata (page 180) tab.</p> <p>Columns (transformations and filters only) (page 154), Headers (page 173), and Metadata (Global transformations only) (page 155) can also be used in the expression and are evaluated before the lookup statement is performed against the endpoint.</p>
'COND'	<p>The condition for the lookup statement.</p> <p>Note: The condition syntax must be native to the endpoint it accesses.</p> <p>The COND is a single field referencing all required fields.</p> <p>Example if the lookup table is located in Oracle:</p> <pre>'Fieldname1=:1 and Fieldname2=:2 and Fieldname3 =:3'</pre> <p>Example if the lookup table is located in Microsoft SQL Server:</p> <pre>'Fieldname1=? and Fieldname2=? and Fieldname3=?'</pre> <p>Columns (transformations and filters only) (page 154), Headers (page 173), and Metadata (Global transformations only) (page 155) can also be used in the expression and are evaluated before the lookup statement is performed against the endpoint.</p>
COND_ PARAMS	<p>Any parameters required by the COND parameter.</p> <p>The COND_PARAMS (condition parameters) is not a single field, but a list of fields.</p> <p>Syntax:</p> <pre>\$FIELDNAME1 , \$FIELDNAME2 , \$FIELDNAME3</pre> <p>Full example:</p> <pre>source_lookup(10000 , 'HR' , 'DEPARTMENTS' , 'DEPARTMENT_NAME' , 'COMPANY_ID=? and DIVISION_ID=? and DEPT_ID=?' , \$COMP_ID , \$DIV_ID , \$DEPT_ID)</pre>

To improve efficiency, the source/target lookup tables should be indexed for the specified lookup fields.

Data Enrichment example

In the following example, Mike needs to add the `DEPARTMENT_NAME` column to the `HR.JOB_HISTORY` table. The `DEPARTMENT_NAME` column is located in the `HR.DEPARTMENTS` table in the source endpoint.

This is how the `HR.JOB_HISTORY` table appears before the column is added:

EMPLOYEE_ID	START_DATE	END_DATE	JOB_ID	DEPARTMENT_ID
102	2001-01-13 00:00:00	2006-07-24 00:00:00	IT_PROG	60
101	1997-09-21 00:00:00	2001-10-27 00:00:00	AC_ACCOUNT	110
101	2001-10-28 00:00:00	2005-03-15 00:00:00	AC_MGR	110
201	2004-02-17 00:00:00	2007-12-19 00:00:00	MK_REP	20
114	2006-03-24 00:00:00	2007-12-31 00:00:00	ST_CLERK	50
122	2007-01-01 00:00:00	2007-12-31 00:00:00	ST_CLERK	50
200	1995-09-17 00:00:00	2001-06-17 00:00:00	AD_ASST	90
176	0001-09-09 09:30:25	2010-09-09 09:30:00	SA_REP	60
176	2007-01-01 00:00:00	2010-09-09 09:30:00	SA_MAN	60
200	2002-07-01 00:00:00	2006-12-31 00:00:00	AC_ACCOUNT	90

This is how the `HR.JOB_HISTORY` table appears after the Full Load completes:

EMPLOYEE_ID	START_DATE	END_DATE	JOB_ID	DEPARTMENT_ID	DEPARTMENT_NAME
102	2001-01-13 00:00:00	2006-07-24 00:00:00	IT_PROG	60	Warehouse
101	1997-09-21 00:00:00	2001-10-27 00:00:00	AC_ACCOUNT	110	ProductManagement
101	2001-10-28 00:00:00	2005-03-15 00:00:00	AC_MGR	110	ProductManagement
201	2004-02-17 00:00:00	2007-12-19 00:00:00	MK_REP	20	HR
114	2006-03-24 00:00:00	2007-12-31 00:00:00	ST_CLERK	50	Bookkeeping
122	2007-01-01 00:00:00	2007-12-31 00:00:00	ST_CLERK	50	Bookkeeping
200	1995-09-17 00:00:00	2001-06-17 00:00:00	AD_ASST	90	Management
176	0001-09-09 09:30:25	2010-09-09 09:30:00	SA_REP	60	Warehouse
176	2007-01-01 00:00:00	2010-09-09 09:30:00	SA_MAN	60	Warehouse
200	2002-07-01 00:00:00	2006-12-31 00:00:00	AC_ACCOUNT	90	Management

To add the `DEPARTMENT_NAME` column, Mike needs to:

1. Create a new task and select the `HR.JOB_HISTORY` table for replication.
2. Apply a “New Column” transformation to the `HR.JOB_HISTORY` table. For more information on defining transformations, see [Defining transformations for a single table/view \(page 93\)](#).
3. Open the **Expression Builder** and choose **Data Enrichment** from the **Functions** tab. For more information on the Expression Builder, see [Using the Expression Builder \(page 149\)](#).
4. Select the `source_lookup` function and configure it as follows (using the native syntax of the source endpoint):

If the lookup table is located in Oracle:

```
source_lookup(10000, 'HR', 'DEPARTMENTS', 'DEPARTMENT_NAME',
'DEPARTMENT_ID=:1', $DEPARTMENT_ID)
```

If the lookup table is located in Microsoft SQL Server:

```
source_lookup
(10000, 'HR', 'DEPARTMENTS', '[DEPARTMENT_NAME]',
'[DEPARTMENT]=?', $DEPARTMENT_ID)
```

Where:

- 10000 is the TTL parameter.
- HR is the schema name.
- DEPARTMENTS is the table name.
- DEPARTMENT_NAME is the expression.
- DEPARTMENT_ID=: 1 (or ? on Microsoft SQL Server) is the condition.
- \$DEPARTMENT_ID is the condition parameter.

5. Run the task.

Operation

The following table describes the Operation functions used by the Expression Builder in Enterprise Manager.

Operation functions

Function	Description
<code>operation_indicator (value_on_delete, value_on_update, value_on_insert)</code>	<p>When the <code>operation_indicator</code> function is invoked on its own or as part of an expression, records deleted from the source table will <i>not</i> be deleted from the corresponding target table. Instead, the corresponding target record will be flagged (with a user-provided value) to indicate that it was deleted from the source. The <code>operation_indicator</code> function also requires you to provide values to indicate records that were inserted or updated in the source endpoint.</p> <div data-bbox="421 577 1390 1077" style="border: 1px solid #ccc; padding: 10px;"> <p> • <i>The <code>operation_indicator</code> function is not supported on tables that do not have a Primary Key.</i></p> <p>• <i>It is recommended to add a dedicated column for the flag values, for example, <code>OPERATION</code>. For an explanation of how to add a column, see Using the Transform tab (page 95).</i></p> <p>• <i>This function is not supported when:</i></p> <ul style="list-style-type: none"> • <i>The Apply Conflicts error handling policy is set to No record found for applying an update: INSERT the missing target record.</i> • <i>The Apply changes using SQL MERGE task setting is enabled.</i> </div> <p>To specify the function values:</p> <p>Replace <code>value_on_delete</code>, <code>value_on_insert</code> and <code>value_on_update</code> with the values that you want to appear in the target endpoint.</p> <p>Values should be formatted according to the corresponding column type.</p> <p>Example when the column type is <code>INT4</code>:</p> <pre>operation_indicator('1', '0', '0')</pre> <p>Example when the column type is <code>STRING</code>:</p> <pre>operation_indicator('Deleted', 'Updated', 'Inserted')</pre>

Other Functions

The following table describes additional functions used by the Expression Builder in Enterprise Manager.

Other functions

Function	Description
length(x)	For a string value x, the length(x) function returns the number of characters (not bytes) in x before to the first NULL character. If x is NULL then length(x) is NULL. If x is numeric then length(X) returns the length of a string representation of X.
like(x,y,z)	The like() function is used to implement the "Y LIKE X [ESCAPE Z]" expression. The ESCAPE (z) clause is optional. If there is a z clause, then the like() function is invoked with three arguments. Otherwise, it is invoked with two arguments.
typeof(x)	The typeof(x) function returns a string that indicates the datatype of the expression x: null, integer, real, text, or BLOB.

Hash

The Hash function generates a hash value for an inputted column (using the SHA-256 algorithm) and then returns the hex value of the generated hash value.

To use the function in an expression, add the `hash_sha256(x)` function to the **Build Expression** pane and then replace the "x" with the desired source column name (from the **Input Columns** tab).

The function is especially useful for masking sensitive information. In the expression below, for example, the Hash function has been used to obfuscate employees' email addresses.

Expression Builder

Input Columns | Header Columns | Operators | Functions

Search

Column	Type
COMMISSION_PCT	NUMERIC(2,2)
DEPARTMENT_ID	INT2
EMAIL	STRING(25)
EMPLOYEE_ID	INT4
FIRST_NAME	STRING(20)
HIRE_DATE	DATETIME(6)
JOB_ID	STRING(10)
LAST_NAME	STRING(25)
MANAGER_ID	INT4
PHONE_NUMBER	STRING(20)
SALARY	NUMERIC(8,2)

1 Build Expression

+ - * / % != == || AND OR

hash_sha256(\$EMAIL)

2 Parse Expression

Parameter	Value
\$EMAIL	<input type="text" value="dan@company.com"/>

3 Test Expression

4D89E44A4ACBF7DE16F09EB5EE0D170B35127104127B168F6B949D296B1F147F

Headers

By default, headers for source tables are not replicated to the target. You can determine which, if any, headers to replicate when you define a transformation by creating an expression that includes the header.

You can create a filter using header values. Header filters are applied during change processing. See [Using filters \(page 103\)](#) for additional information.



- The **Headers** tab in the Expression builder is available for filters and transformations. It is available for Global transformations only when you select **Add Columns**. See [Transformation type \(page 120\)](#).
- Header columns are supported by all endpoints, except where explicitly stated to the contrary.

The available headers are described below.

AR_H_CHANGE_SEQ

Value in Change Processing

A monotonically increasing change sequencer that is common to all Change tables of a task. The Change sequence has the following format (with time being the UTC time on Replicate Server):

YYYYMMDDHHmmSShhxxxxxxxxxxxxxxxxxxxxxx

Where:

- YYYY is the four-digit year (such as 2012)
- MM is the two-digit month (range from 01-12)
- HH is the hour in the day (range from 00-23)
- mm is the minute in the hour (range from 00-59)
- SS is the second in the minute (range from 00-59)
- hh is the hundredth of the second (range from 00-99)
- xxxxxxxxxxxxxxxxxxx is a 19-digit, zero prefixed change number (global per task).

The time part usually refers to the commit time of the transaction that includes the change record. Qlik Replicate contains logic that maintains the monotonicity of the sequence number so modifying or adjusting the endpoint time may result in multiple changes to seem that they are within the same timestamp but with increasing change number.

The xxx...xxx is usually the internal change number from the data record except that for BEFORE-IMAGE records it is the same as the change number of the matching UPDATE record (for example, if the change number of BEFORE-IMAGE is 1000 and that of the UPDATE is 1001, then both have 1001). This allows a simple left-outer-join between the table and itself where on the left we scan until the point in time but filter out operation=before-image, and on the right we join on the same change_seq with the change_oper being 'B'.

Value in Full Load

Empty

Data type

VARCHAR(35)

AR_H_STREAM_POSITION

Value in Change Processing

The stream position value on the source - usually the SCN or LSN - which allows Replicate tasks to resume from the last processed event.

Value in Full Load

Empty string

Data type

STRING

AR_H_TIMESTAMP

Value in Change Processing

The timestamp of the Change operation, which is represented as the local time on Replicate Server. If your database and Replicate Server are located in the same timezone, the timestamp will be approximately representative of the actual Change operation.

Value in Full Load

Current timestamp

Data type

DATETIME

AR_H_TRANSACTION_ID**Value in Change Processing**

The ID of the transaction to which the Change record belongs. The actual value is a hex-string of the 128-bit transaction ID. Depending on the source endpoint type, the ID may either be similar to the transaction ID in the source database or identical to it.



The transaction ID is not guaranteed to be unique and therefore should not be used to create reports or any other operation that relies upon its uniqueness.

Value in Full Load

Empty

Data type

VARCHAR (32)

AR_H_COMMIT_TIMESTAMP**Value in Change Processing**

The source database commit timestamp, according to Replicate Server time. When the source database and Replicate Server are in the same timezone, the timestamp will be the approximate time of the actual commit on the source database.

Value in Full Load

Current timestamp

Data type

DATETIME

AR_H_DB_COMMIT_TIMESTAMP**Value in Change Processing**

The source database commit timestamp, according to the database server time.



Relevant for the following source endpoints only: Oracle, Microsoft SQL Server, IBM DB2 for z/OS, Microsoft Azure SQL Managed Instance, and Amazon RDS for SQL Server.

Value in Full Load

NULL

Data type

DATETIME

AR_H_OPERATION

Value in Change Processing

Can be one of the following: INSERT, UPDATE, or DELETE.

Value in Full Load

INSERT

Data type

STRING

AR_H_USER

Value in Change Processing

The user that made the change to the source database.



Relevant for the following source endpoints only:

- *Microsoft SQL Server*
- *Microsoft Azure SQL Managed Instance*
- *Amazon RDS for SQL Server*
- *IBM Informix*
- *IBM DB2 for z/OS*
- *IBM DB2 for LUW*
- *Oracle*
- *File*
- *SAP Sybase ASE*

Value in Full Load

Empty

Data type

STRING

AR_H_JOB_NAME

Value in Change Processing

The iSeries job that made the change to the source database.



Relevant for the IBM DB2 for iSeries source endpoint only.

Value in Full Load

Empty

Data type

STRING

AR_H_PROGRAM_NAME

Value in Change Processing

The iSeries program that made the change to the source database.



Relevant for the IBM DB2 for iSeries source endpoint only.

Value in Full Load

Empty

Data type

STRING

AR_H_XACT_ID

Value in Change Processing

As opposed to the TRANSACTION_ID which is retrieved from the physical LDF TLOG file, the XACT ID is the transaction ID available during the MS SQL transaction itself. The ID is the Log Sequence Number (LSN) of the first record for the last distributed transaction of the server. All records in the transaction will have the same ID.



Relevant for the following source endpoints only:

- *Microsoft SQL Server*
- *Microsoft Azure SQL Managed Instance*
- *Amazon RDS for SQL Server*

Value in Full Load

Empty

Data type

VARCHAR (32)

User-defined transformations

Customers that requires functionality not provided by Replicate's built-in transformations can write their own transformations, and then access them from the [Replicate Expression Builder](#).

It is also recommended to use the Replicate Add-ons API which allows you to take advantage of Replicate's memory management and logging capabilities, while eliminating the need to create independent memory and logging routines.

The procedure below is based on the sample files located in:

```
<INSTALL_DIR>\addons\samples\MyTransformation
```



The path is the same on Linux, but with slashes (/) instead of backslashes (\). Similarly, although the explanation below refers to a DLL file (Windows), on Linux this should be an SO file.

Changing the default addon name

You can change the default addon name (**MyTransformation**) simply by renaming the `<INSTALL_DIR>\addons\samples\MyTransformation\MyTransformation.dll` file as desired. Note that if you rename the DLL, you may also need to specify the new name when editing the `addons_def.json.sample` file described below (depending on which parameters you use).

Creating a user-defined transformation

To create a user-defined transformation:

1. Create a shared library that implements the following exported initialization function:

```
typedef int AR_ADDON_INIT_FUNC(AR_ADDON_CONTEXT *context);
```

All of the types and prototypes are defined in the `ar_addon.h` and `ar_addon_transformation.h` files located under `<INSTALL_DIR>\addons\include`.
2. Make sure your shared library is in the following location:
`<INSTALL_DIR>\addons\samples\addon_name`



When setting up Qlik Replicate in a Cluster Environment, the created binaries should be installed on all of the cluster instances.

3. Compile the transformation.
The DLL is automatically created in the following location:
`<installation_dir>\addons\addon_name\addon_name.dll`
4. Register the library in the `addons_def.json.sample` file located under `<INSTALL_DIR>\addons`.

```
{
  "addons": [{
    "name": "MyTransformation",
    "type": "STARTUP",
    //"lib_path": "C:\\Program
Files\\Attunity\\Replicate
\\addons\\samples\\MyTransformation\\MyTransformation.dll",
    //"lib_path":
"/opt/attunity/replicate/addons/samples/MyTransformation/MyTransformatio
n.so",
    "init_function": "my_transformation_init_func"
  }]
}
```

Where:

- `name` is the logical name of the DLL (can be any name). If you do not use the `lib_path` parameter to specify the DLL path (see below), then the DLL file must reside in `<INSTALL_DIR>\addons\<addon_name>`. On Linux, the `<addon_name>` folder needs to be created manually and should either be the default addon name (**MyTransformation**) or its new name (if you changed it).
- `type` is an optional parameter that specifies when to load the DLL. The only value that is currently supported is `STARTUP`.
- `lib_path` is the full path of your DLL (e.g. `C:\Transformations\MyManipulator.dll`). This is required only if the DLL does not reside in `<INSTALL_DIR>\addons\<addon_name>` (as mentioned in the description of the `name` parameter above).
- `init_function` is the function name, as it appears in the C file used to generate the DLL.

5. Save the file as **addons_def.json**.

6. Register the new function in the addon initialization function (mentioned in Step 1) as in the following example:

```
USER_DEFINED_TRANSFORMATION_DEF *transdef = GET_AR_AO_TRANSFORMATION_DEF
();
transdef->displayName = "prefix_with(X, Y)";
transdef->functionName = "prefix_with";
transdef->description = "prefix_with adds the prefix <Y_> to a given
string X";
transdef->func = trans_prefix_with;
transdef->nArgs = 2;
AR_AO_REGISTRATION->register_user_defined_transformation(transdef);
```

7. Restart the Qlik Replicate Server service.

The new "prefix_with" function will be available in the Expression Builder under **Functions->User Defined**.

9.4 Task Settings

Task-specific replication settings can be configured in the **<Task Name> - Settings** dialog box.



Some of the task settings are not available in a Log Stream Staging setup.

For information on the availability of task settings in a Log Stream Staging setup, refer to the Qlik Replicate Setup and User Guide.

To open the <Task Name> - Settings dialog box:

1. Open the desired task.
For information on opening a task, see [Editing a replication task \(page 82\)](#).
2. Click the **Task Settings** toolbar button.
The **<Task Name> - Settings** dialog box opens, displaying the following tabs:

- [Metadata \(page 180\)](#)
 - [Target metadata \(page 180\)](#)
 - [Control tables \(page 184\)](#)
- [Bidirectional \(page 186\)](#)
- [Full Load \(page 187\)](#)
 - [Full Load Settings \(page 187\)](#)
 - [Full Load Tuning \(page 189\)](#)
- [Change processing \(page 190\)](#)
 - [Apply Changes Settings \(page 190\)](#)
 - [Store Changes Settings \(page 191\)](#)
 - [Change Processing Tuning \(page 197\)](#)
- [Error handling \(page 203\)](#)
 - [Error Handling settings \(page 204\)](#)
 - [Environmental errors \(page 204\)](#)
 - [Data Errors \(page 205\)](#)
 - [Table Errors \(page 208\)](#)
 - [Apply Conflicts \(page 208\)](#)
- [Logging \(page 210\)](#)
- [Character substitution \(page 211\)](#)
- [File uploads \(page 213\)](#)
- [Message format \(page 214\)](#)
- [Transformations and filters \(page 219\)](#)
- [More options \(page 220\)](#)

Metadata

Clicking the **Metadata** tab in the **Task Settings** window reveals the following sub-tabs:

- [Target metadata \(page 180\)](#)
- [Control tables \(page 184\)](#)

Target metadata

Target table schema: The schema on the target to which the source tables will be replicated if you do not want to use the source table schema (or if there is no schema in the source database).



When replicating to Hadoop-based targets, the value specified in this field will be interpreted as a database name (as opposed to a schema name).

LOB handling options

For information on how to override these settings for individual tables, see [Handling LOB columns \(page 114\)](#).



- During CDC or during Full Load when the **Allow unlimited LOB size** option is enabled, LOB data types are supported only in tables with a primary key or unique index.
- When replicating from Microsoft SQL Server, inline LOBS will always be read directly from the logs (i.e. without lookup).

The following LOB handling options are available:

LOB handling options

Option	Description
Replicate LOB columns	<p>When this option is selected (the default), LOB columns will be replicated.</p> <p>Note that replicating LOBs may impact performance. This is especially true in the case of the large LOBs which require Replicate to perform a lookup from the source table in order to retrieve the source LOB value.</p> <div style="border: 1px solid #ccc; padding: 10px; margin-top: 10px;"> <i>Enabling Replicate LOB columns while a task is running will not be reflected in the target tables, unless the target tables are recreated by Qlik Replicate. This should be done as follows:</i> <ol style="list-style-type: none"> a. Make sure that DROP and CREATE change table in the task settings' Store Changes Settings tab is selected (the default). b. Enable Replicate LOB columns and then reload the target tables. </div>
Allow unlimited LOB size	<p>Select this option - also known as Full LOB mode - to ensure that all LOBs are replicated without being truncated. This option should be selected when all (or nearly all) of the LOBs you wish to replicate are large (i.e. exceed 1 GB).</p> <div style="border: 1px solid #ccc; padding: 10px; margin-top: 10px;"> Note <i>If the task's Change Processing Mode is set to "Batch optimized apply" (the default), Replicate will switch to "Transactional apply" mode to apply tables with LOBs.</i> </div>

Option	Description
Optimize handling when LOB size is less than (KB)	<p>Select this option when you need to replicate both small and large LOBs, and most of the LOBs are small.</p> <div data-bbox="376 389 1388 667" style="border: 1px solid #ccc; padding: 10px; margin: 10px 0;"> <p> <i>This option is supported with the following endpoints only:</i></p> <ul style="list-style-type: none"> • Sources: Oracle, Microsoft SQL server, MySQL, PostgreSQL, IBM DB2 for LUW, and Sybase ASE. • Targets: Oracle, Microsoft SQL Server, MySQL, PostgreSQL, IBM DB2 for z/OS, and Sybase ASE. </div> <p>When this option is selected, during Full Load, the small LOBs will be replicated "inline" (which is more efficient), and the large LOBs will be replicated by performing a lookup from the source table.</p> <p>When capturing changes, however, both small and large LOBs will be replicated by performing a lookup from the source table.</p> <div data-bbox="376 936 1388 1205" style="border: 1px solid #ccc; padding: 10px; margin: 10px 0;"> <p> <i>When this option is selected, Replicate will check all of the LOB sizes to determine which ones to transfer "inline". LOBs larger than the specified size will be replicated using Full LOB mode.</i></p> <p><i>Therefore, if you know that most of the LOBs are larger than the specified setting, it is better to use the Allow unlimited LOB size option instead.</i></p> </div>
Chunk size (KB)	<p>Optionally, change the size of the LOB chunks to use when replicating the data to the target. The default chunk size should suffice in most cases, but if you encounter performance issues, adjusting the size may improve performance.</p> <div data-bbox="376 1368 1388 1570" style="border: 1px solid #ccc; padding: 10px; margin: 10px 0;"> <p> <i>With some databases, data type validation occurs when the data is inserted or updated. In such cases, replication of structured data types (e.g. XML, JSON, GEOGRAPHY, etc.) may fail if the data is bigger than the specified chunk size.</i></p> </div>

Option	Description
Limit LOB size to (KB)	<p>Select this option if you only need to replicate small LOBs or if the target endpoint does not support unlimited LOB size. The maximum permitted value for this field is 102400 KB (100 MB).</p> <p>When replicating small LOBs, this option is more efficient than the Allow unlimited LOB size option since the LOBs are replicated "inline" as opposed to via "lookup" from the source. During Change Processing, small LOBs are usually replicated via "lookup" from the source.</p> <p>The limit applies to the number of LOB bytes that is read from the source endpoint. For BLOB columns, the target BLOB size will be exactly according to the specified limit. For CLOB and NCLOB columns, the target LOB size might differ from the specified limit if the source and target LOB do not have the same character set. In this case, Replicate will perform character set conversion, which might result in a discrepancy between the source and target LOB sizes.</p> <p>As the value of the Limit LOB size to is in bytes, the size should be calculated according to the following formulas:</p> <ul style="list-style-type: none"> • BLOB - The length of the largest LOB. • NCLOB - The length of the longest TEXT <i>in characters</i> multiplied by two (as each character is handled as a double-byte). If the data includes 4-byte characters, multiply it by four. • CLOB - The length of the longest TEXT in characters (as each character is handled as a UTF8 character). If the data includes 4-byte characters, multiply it by two. <div style="border: 1px solid #ccc; padding: 10px; margin-top: 10px;"> <p> • <i>Any LOBs larger than the specified size will be truncated.</i></p> <p>• <i>When capturing changes from Oracle source, inline BLOBs are replicated inline.</i></p> <p>• <i>Changes to this setting will only affect existing tables after they are reloaded.</i></p> </div>



In some scenarios, tasks configured to replicate tables with multiple LOB columns may consume a large amount of memory. This is because Replicate allocates memory by multiplying the **Limit LOB size to** value by the [Commit rate during full load](#) value, the sum of which, it multiplies by the number of LOB columns being replicated. So, for example, if LOB size is limited to 5 MB and the default commit rate is used (10000 events), a task replicating 6 LOB columns will consume 300 GB of memory. Note that other factors such as the database type and version may also affect memory consumption.

Should you encounter memory consumption issues and suspect that a combination of the above factors may be the cause, stop the task and lower the value in the [Commit rate during full load](#) field. Then resume the task. Repeat this process until acceptable performance/memory levels are reached.

These instructions apply to Change Processing and Full Load tasks.



Changes to a column's LOB size while a task is running will not be reflected in the Change Table, unless the target tables are created by Qlik Replicate. In such cases, the task must be configured to drop and create the Change Table (the default) and the target tables need to be reloaded (after the LOB size has changed).

For more information on the Change Table, see [Store Changes Settings \(page 191\)](#). For information on reloading target tables, see the [Qlik Replicate Help](#).

Control tables

Control Tables provide information about the replication task as well as useful statistics that can be used to plan and manage both the current replication task and future replication tasks. Aside from the **Apply Exceptions** table which is always created, you can [choose which Control Tables to create on the target](#).

Create target control tables in schema: Enter the endpoint schema for the target Control Tables. If you do not enter any information in this field, then the tables will be created in the default location in the endpoint.



When this field is left empty, the target endpoint is MySQL, and the Multiple Endpoints option is enabled, a default database named `attrep_control` will be created on the MySQL server. The selected control tables will be created in this database.

For more information on the Multiple Endpoints option, see the [Qlik Replicate User and Reference Guide](#).



When replicating to a Hadoop target endpoint, the value specified in this field will be interpreted as a database name (as opposed to a schema name).

Create target control tables in tablespace: When the replication target is Oracle, specify the tablespace where you want the target control tables to be created. If you do not enter any information in this field, the tables will be created in the default tablespace in the target database.

Create target control table indexes in tablespace: When the replication target is Oracle, specify the tablespace where you want the control table indexes to be created. If you do not enter any information in this field, the indexes will be created in the same tablespace as the control tables.

Replication history time slot (minutes): The length of each time slot in the Replication History table. The default is 5 minutes.

Table selection

In addition to the **Apply Exceptions** table (required), select which of the following Control Tables you want Qlik Replicate to create on the target endpoint and set the **Update every** and **Insert only** options as needed:

- **Replication Status:** Provides details about the current task including task status, amount of memory consumed by the task, number of changes not yet applied to the target and the position in the source endpoint from which Qlik Replicate is currently reading.
 - **Update every:** The **Replication Status** table update frequency. The default is 5 minutes. A higher value might help reduce the possibility of locks due to numerous tasks attempting to update the table simultaneously.
 - **Insert only:** When this is enabled, Replicate will insert records to the `attrep_status` control table instead of updating existing records. This is useful for preventing table locks on targets such as Snowflake that limit the number of concurrent UPDATE operations on the same table.
- **Suspended Tables:** Provides a list of suspended tables as well as the reason they were suspended.
- **Replication History:** Provides information about the replication history including the number and volume of records processed during a replication task, latency at the end of a CDC task, among others.
 - **Update every:** The **Replication History** table update frequency. The default is 5 minutes. A higher value might help reduce the possibility of locks due to numerous tasks attempting to update the table simultaneously.
- **Change Data Partitions:** The `attrep_cdc_partitions` table contains records of partitions created on the target database when [Change Data Partitioning \(page 193\)](#) is enabled for a Replicate task. You can use this information to identify partitioned data that needs to be further processed.
- **DDL History:** The `attrep_ddl_history` table contains a history of all supported DDL changes that occurred during a task.

For a list of DDL changes supported by Replicate, refer to the Qlik Replicate Setup and User Guide. Note that DDL changes written to this Control Table are also subject to the limitations described in the section in the "Limitations when Capturing DDL Changes" section in the Qlik Replicate Setup and User Guide.



The DDL History table is currently supported with the following target endpoints only:

- Amazon EMR
- Amazon Kinesis Data Streams
- Amazon Redshift
- Amazon S3
- Amazon MSK
- Cloudera Data Platform (CDP) Private Cloud
- Confluent Cloud
- Databricks (Cloud Storage)
- Databricks Lakehouse (Delta)
- File
- Google Cloud Pub/Sub
- Google Cloud Storage
- Google Dataproc
- Hadoop
- Hortonworks Data Platform
- Kafka
- Microsoft Azure ADLS
- Microsoft Azure Event Hubs
- Microsoft Azure HDInsight
- Microsoft Fabric Data Warehouse
- Snowflake on Google
- Snowflake on AWS
- Snowflake on Azure

For a detailed description of these tables, see the Qlik Replicate User and Reference Guide.

Bidirectional

This tab is only applicable to bidirectional replication tasks. When you click **Bidirectional** in the **Task Settings** dialog box, the **Loopback Prevention** tab is displayed. In bidirectional replication, loopback prevention is a mechanism that prevents the same data from being replicated back and forth in an endless loop. To enable loopback prevention, you need to specify a source and target **Loopback prevention table schema**.

Bidirectional replication consists of two separate tasks: Task 1 captures changes made to Endpoint A and replicates them to Endpoint B. Task 2 captures changes made to Endpoint B and replicates them to Endpoint A. When configuring Task 1 of a bidirectional replication setup, the source loopback prevention table schema must be identical to the target loopback prevention table schema specified in the **Loopback Prevention** settings of Task 2.

Likewise, when configuring Task 2 of a bidirectional replication setup, the source loopback prevention table schema must be identical to the target loopback prevention table schema specified in the **Loopback Prevention** settings of Task 1.



*Oracle schemas are case-sensitive. Therefore, when specifying an Oracle table schema, make sure to use the correct case in the **Loopback Prevention** settings in both Tasks.*

For instructions on setting up bidirectional replication, see [Bidirectional replication \(page 71\)](#).

Full Load

Clicking the **Full Load** tab in the **Task Settings** window reveals the following sub-tabs:

- [Full Load Settings \(page 187\)](#)
- [Full Load Tuning \(page 189\)](#)

Full Load Settings

Click the **Full Load Settings** sub-tab to configure the following:

Full Load Processing is ON/OFF.

Click this button to toggle full load on or off. The initial setting is determined when [Adding tasks \(page 69\)](#).

When full load is ON, Qlik Replicate loads the initial source data to the target endpoint.



Full load can be turned on or off at any stage even if change processing is on. Once the task begins to process changes, the full load on/off switch is used only as additional protection against accidental or unauthorized reload.

Target table preparation

If target table already exists: Select one of the following from the list to determine how you want to handle loading the target at full-load start up:



The option to drop or truncate the target tables is relevant only if such operations are supported by the source endpoint.

- **DROP and Create table:** The table is dropped and a new table is created in its place.



Replicate Control Tables will not be dropped. However, any suspended tables that are dropped will also be deleted from the `attrep_suspended_tables` Control Table if the associated task is reloaded.

- **TRUNCATE before loading:** Data is truncated without affecting the table metadata. Note that when this option is selected, enabling the [Create primary key or unique index after full load completes](#) option will have no effect.
- **ARCHIVE and CREATE table:** A copy of the existing table will be saved to the same schema before the new table is created. The archived table name will be appended with a timestamp, indicating when the archiving operation occurred (e.g. Customers_20170605175601).



Currently this option is only available for the Hadoop target endpoint .

- **Do nothing:** Existing data and metadata of the target table will not be affected. New data will be added to the table.



*Replicate expects the source column data types to be compatible with the corresponding target column data types. If you choose either **TRUNCATE before loading** or **Do nothing** and one or more target data types are different than the data types for the corresponding source columns, use a transformation to convert the data types as required.*

For information on creating data type transformations, see [Defining transformations for a single table/view \(page 93\)](#).

Primary Key or Unique Index Creation

Create primary key or unique index after full load completes: Select this option if you want to delay primary key or unique index creation on the target until after full load completes.

Stopping the Task after Full Load

After Full Load completes, stop the task: You can set the task to stop automatically after Full Load completes. This is useful if you need to perform DBA operations on the target tables before the task's Apply Changes (i.e. CDC) phase begins.

During Full Load, any DML operations executed on the source tables are cached. When Full Load completes, the cached changes are automatically applied to the target tables (as long as the **Before/After cached changes have been applied** option(s) described below are disabled).



This feature is not available for bidirectional replication tasks.

Select **Before cached changes have been applied** to stop the task before the cached changes are applied and/or **After cached changes have been applied** to stop the task after the cached changes are applied.

Selecting the **Before cached changes have been applied** option will stop the task after Full Load completes. Selecting the **After cached changes have been applied** option will stop the task as soon as data is consistent across all tables in the task.



When configuring Replicate to stop the task after Full Load completes, note the following:

- The task does not stop the moment Full Load completes. It will be stopped only after the first batch of changes has been captured (as this is what triggers the task to stop). This might take a while depending on how frequently the source database is updated. After the task stops, the changes will not be applied to the target until the task is resumed.
- The task will stop after Full Load completes, even if there are no cached changes to apply.
- The **After cached changes have been applied** option is not supported with all file-based and Hadoop-based target endpoints, namely:
 - **File-based:** File, Amazon S3, Microsoft Azure ADLS, and Google Storage.
 - **Hadoop-based:** Hadoop, Hortonworks Data Platform, Amazon EMR, Microsoft Azure HDInsight, Google Dataproc, Cloudera Data Platform (CDP) Private Cloud, and Microsoft Azure Databricks.
- Choosing to stop the task before cached changes have been applied may adversely affect performance, since the cached changes will only be applied to tables (even those that have already completed Full Load) after the last table completes Full Load.
- When the **Before/After cached changes have been applied** option is selected and a DDL is executed on one of the source tables during the Full Load process (in a Full Load and Apply Changes task), Replicate will reload the table. This effectively means that any DML operations executed on the source tables will be replicated to the target before the task stops.
- When working with the File Channel endpoint, these options should be set in the remote File Channel task and not in the local File Channel task.
For more information on the File Channel endpoint, see the *Qlik Replicate Setup and User Guide*.

Duplicate Record Prevention



Supported when using the IBM DB2 for z/OS and IBM DB2 for iSeries source endpoints only.

Select the **Eliminate creation of duplicate records on full load** option if you need to prevent duplicate records from being replicated during Full Load. You can either set the option at task level or per table.

Note that selecting this option could impact performance as Replicate instructs the source database to return the table records by Primary Key order and then removes any duplicate records.

For information on preventing creation of duplicate records at table level, see [Full Load \(page 119\)](#).

Full Load Tuning

Click the **Full Load Tuning** sub-tab to configure the following:

Tuning settings:

- **Maximum number of tables to load in parallel:** Enter the maximum number of tables to load into the target at one time. The default value is **5**.
- **Transaction consistency timeout (seconds):** Enter the number of seconds that Qlik Replicate waits for transactions to close, if they are open when the task starts, before beginning the Full Load operation. The default value is **600** (10 minutes). Qlik Replicate will begin the full load after the timeout value is reached even if there are open transactions.



This setting is not applicable to Microsoft SQL Server.



To replicate transactions that were open when Full Load started but were only committed after the timeout value was reached, you need to reload the target tables.

- **Commit rate during full load:** The maximum number of events that can be transferred together. The default value is **10000**.

Change processing

Clicking the **Change Processing** tab in the **Task Settings** window reveals the following sub-tabs:

- [Apply Changes Settings \(page 190\)](#)
- [Store Changes Settings \(page 191\)](#)
- [Change Processing Tuning \(page 197\)](#)

Apply Changes Settings

Click the **Apply Changes Settings** sub-tab to configure the following:

Apply Changes is ON/OFF:

Click this button to toggle Apply Changes (Change Processing) on or off. The initial setting is determined when [Adding tasks \(page 69\)](#).

When Apply Changes is ON, Qlik Replicate processes the changes. You can view the change processing in the Monitor. For more information, see the Qlik Replicate Setup and User Guide.



When you turn on apply changes you must reload the task or position back to the point of the previous reload.

DDL handling policy: Determine how to handle the target table for the change capture:



- Executing a DDL on a source table during the Full Load process in a Full Load and Apply Changes task will cause Replicate to reload the table.
- The option to drop or truncate the target tables is relevant only if such operations are supported by the source endpoint.
- Executing the Rename Table DDL with "Ignore ALTER" selected will cause events to continue being delivered until you stop and then resume the task.

- When source table is dropped, select one of the following:
 - **DROP target table**
 - **Ignore DROP**
- When source table is truncated, select one of the following:
 - **TRUNCATE target table**
 - **Ignore TRUNCATE**
- When source table is altered, select one of the following:
 - **ALTER target table**
 - **Ignore ALTER**

Store Changes Settings

When **Store Changes** is turned on in the task settings, you can configure the **Store Changes Settings** for a replication task.

Store changes processing is ON/OFF:

Click this button to toggle **Store Changes** on or off. The initial setting is determined when [Adding tasks \(page 69\)](#). If this option is ON, changes are stored in either Change Tables or an Audit Table.

For more information about storing and applying changes, see the Qlik Replicate Setup and User Guide.



Store Changes can be turned on or off at any time without affecting anything in the task. Changes that are processed and not stored as a result of change storage being turned off can be recovered only by setting the task to an earlier point in time.

If Store Changes is **ON**, use the following options to determine how to store changes. Changes can be stored in Change Tables or in a single Audit table. From the **Store changes in** drop-down list, choose either **Change tables** or **Audit table** according to your needs.

Storing changes in Change Tables

The following section describes the options that are available when storing changes in Change Tables.

- **Suffix:** Type a string to use as the suffix for all Change Tables. The default value is `__ct`. The Change Table names are the name of the target table with the suffix appended. For example, if you have a table called HR and use the default value, the name of the Change Table will be `HR__ct`. For more information, see the Qlik Replicate Setup and User Guide.

- **Header column prefix:** Type a string to use as the prefix for all of the Change Table header columns. The default value is `header__`.
For example, the header column `stream_position` when using the default value is called `header__stream_position`.
For more information, see the Qlik Replicate Setup and User Guide.
- **DDL options:** Select one of the following options to determine how to handle DDL operations on the source tables:
 - **Apply to change table** - DDL operations to the source tables (such as a column being added) will be applied to the corresponding Replicate Change Tables only.
 - **Ignore** - All DDL operations to source tables will be ignored.
 - **Apply to change table and target table**
This is the default option. When this option is selected, DDL operations to the source tables will be applied both to the Replicate Change Tables and to the corresponding target tables.
- **On UPDATE:** Select one of the following options to determine how to store UPDATES to the source tables:
 - **Store before and after image** - To store both the pre-UPDATE data and the post-UPDATE data.
 - **Store after image only** - To store only the post-UPDATE data.

Change table creation:

If Change Table exists when full load starts: Select one of the following to determine how you want to handle loading the Change Tables when Full Load replication starts:

- **DROP and CREATE Change Table:** The table is dropped and a new table is created in its place.
- **ARCHIVE and CREATE Change Table:** A copy of the existing table will be saved to the same schema before the new table is created. The archived table name will be appended with a timestamp, indicating when the archiving operation occurred (e.g. `customers__ct_20170605175601`).



Currently this option is only available for the Hadoop target endpoint.

- **Delete old changes and store new changes in existing Change Table:** Data is truncated and added without affecting the table metadata.
- **Keep old changes and store new changes in existing Change Table:** Data and metadata of the existing Change table are not affected.

Change Data Partitioning



This feature is supported with the following target endpoints only:

- *Hadoop (Hortonworks and Cloudera)*
- *File*
- *Amazon S3*
- *Databricks (Cloud Storage)*
- *Microsoft Azure HDInsight*
- *Google Cloud Storage*
- *Microsoft Azure ADLS*
- *Cloudera Data Platform (CDP) Private Cloud*
- *Google Data Proc*
- *Amazon EMR*
- *Hortonworks Data Platform (HDP)*

In a standard replication task, changes are replicated to the target in no particular order. Change Data Partitioning enables processing of Change Data from many tables in a consistent fashion. You can define the duration of partitions as well as the partitioning base time, thereby ensuring overall consistency of the partitioned data (i.e. no partial transactions, no order headers without order lines, and so on.)

The partitioned data is stored in the Replicate Change Tables. When the Change Data Partitions table is selected (in the [Control tables \(page 184\)](#) tab), information about the partitions will be recorded in the `attrep_cdc_partitions` Control Table on the target database. This information can be used to identify partitioned data that needs to be further processed.

The partitioning options are as follows:

- **Off** - Replicate Change Data without partitioning.
- **Partition every** - Specify the length (in hours and minutes) of each partition.



It is recommended to specify a partition length in excess of one hour. Although specifying a partition length less than one hour may improve latency, creating many partitions on the target may also impact (target) performance (especially in systems with large volumes of changes).

If you resume a task from BEFORE the time that the last partition was created, Replicate will write to a partition that has already been closed.

- **Partition base time** - Partitions are created during a 24 hour time period, which is calculated according to the specified “Partitioning base time” on the source database (in UTC time). For example, a partition interval of 8 hours with a “Partitioning base time” time of 02:00 will create the following partitions: 02:00-10:00, 10:00-18:00, 18:00-02:00 - but not necessarily in that order. For instance, if a task started at 01:00, then the timeframe of the first partition will be 18:00-02:00.

Additionally, if a task started in the middle of a partition (e.g. at 04:00), its Change Data will be inserted into the 02:00-10:00 partition (even though no changes were captured before 04:00).

- **Speed partition mode**



- *This feature is supported with Hadoop-based target endpoints only.*
- *This feature cannot be enabled together with [Optimize file uploads](#).*

By default, Change Data Partitions for all tables in a replication task are registered on the target at the same time. As soon as the partitions are registered, information about them (such as a partition's start and end time) is also published to the Replicate Change Data Partitions Control Table. This ensures data consistency across all of the replicated tables.

In **Speed partition mode**, rather than waiting for all Change Data files to be uploaded (for all tables), Replicate creates and registers a partition per-table as soon as the partition's first data file is uploaded (to its table). Creating and registering partitions *before* all the Change Data has been uploaded is especially useful for consuming applications such as Qlik Compose that need to process changes with a minimum of delay, as the Change Data rapidly becomes available for consumption, even if it is incomplete.

For example, if an endpoint is configured to upload Change Data files every five minutes, each batch of changes can be processed immediately, even if the Partition every interval is set to six hours.

However, despite significantly reducing latency, working in this mode may have some possible drawbacks:

- Accumulation of a large number of files per partition, which may impact performance on the target.
It is therefore strongly recommended to clean up old partitions using the Partition Retention options (see below) provided for this purpose.
- Inconsistent results may be encountered when querying several tables due to the Change Data arriving on the target at different times.



Change Tables that were created before Change Data Partitioning was enabled need to be dropped or renamed so that they can be recreated with the [additional "partition_name" column](#).

Deleting old partitions periodically

Over time, Change Data Partitioning can result in a large number of files and partitions accumulating on the target system, which may significantly impact performance. Therefore, to ensure optimal performance, best practice is to delete old partitions from time to time.

There are two ways of deleting processed partitions: periodically and ad-hoc. This topic explains how to set up periodic partition deletion. For information on ad-hoc partition deletion, refer to "Deleting Old Partitions Manually" in the Replicate Help.



Currently, this feature is supported with the Microsoft Azure Databricks endpoint only.

Setting up periodic deletion is a two-phase process which involves specifying a deletion interval in the UI, and setting a partition retention barrier using the Enterprise Manager API.

To do this:

1. Enable the **Partition Retention** option.
Any partitions created while this option is disabled will not be deleted when deletion is initiated (either periodic or manual). Therefore, the **Partition Retention** option should only be disabled if you are absolutely certain that periodic deletion will not be initiated at any point in the future.
2. Specify a partition deletion interval in **Day**, **Hours** or **Minutes**.
To prevent the deletion of open partitions or recently closed partitions, the partition deletion interval must be at least double the **Partition every** value.
3. Set a retention barrier date by calling the relevant Enterprise Manager API method.
To ensure that only old (i.e. processed) partitions will be deleted, periodic deletion requires a retention barrier date to be set. The retention barrier date should be set by the consuming application each time it finishes processing a partition. A consuming application can be any application (Qlik or otherwise) tasked with processing the partitioned data. There may be several consuming applications, each of which sets its own retention barrier. In this case, partitions will be deleted up to the earliest retention barrier. For example, if Application A sets July 7th, 2020 as a barrier, Application B sets August 7th, 2020 as a barrier, and Application C sets September 7th, 2020 as a barrier, partitions will be deleted up to July 7th, 2020.
As soon as a retention barrier is set, periodic deletion will start to run according to the defined interval, until any of the following occurs:
 - The **Partition Retention** option is disabled
 - An API call removes the retention barrier
 - All partitions until the retention barrier have been deleted

For more information on setting and removing a partition retention barrier using the API, refer to the Enterprise Manager Developer's Guide.

Limitations and considerations

- If a connection error occurs during the partition deletion operation, the operation will fail (with a warning). The operation will continue from the point of failure the next time that deletion is performed (either periodically or manually).
- If a specific object (partition or storage) cannot be deleted due to other errors (i.e. not connection errors), a warning will be written to the log and deletion of the remaining objects will continue. Partitions that cannot be deleted due to such errors must be deleted directly from the database by the DBA.
- To prevent deletion of open partitions, the last partition in a table will not be deleted, even if meets the criteria for deletion.
- Deletion of partitions in renamed tables is not supported.
- Reloading a target with the [Drop and Create tables](#) option (the default) will delete both the tables and their partitions, regardless of any partition deletion policy. However, reloading a target with the **Do nothing** option, will leave the partitions untouched. In this case, partitions created prior to the target reload cannot be deleted using the Replicate partition deletion tools.

Selecting Change Table header columns

The Change Table header columns provide information about the Change Processing operation such as the type of operation (e.g. INSERT), the commit time, and so on. If you do not need this information, you can configure Replicate to create the Change Tables without some or all of the header columns, thereby reducing their footprint in the target database. To do this, clear the check boxes next to the header columns that you wish to exclude.

Note that you cannot remove additional columns or restore columns while a task is running. To change your initial selection, you first need to stop the task, then modify your selection, and finally reload the target tables.



When [Change Data Partitioning](#) is enabled, an extra header column named "partition_name" is added to the Change Tables and automatically selected in the UI. As this column is required, it cannot be excluded.

For a description of the header columns, refer to the Qlik Replicate User Guide and Reference .

Storing changes in an Audit table

The following section describes the options that are available for storing changes in an Audit table.



- If your source tables contain records exceeding 4096 characters, to prevent truncation of these records, you must turn on the Task Settings' [Replicate LOB columns](#) option (even if the source tables do not contain any LOB columns).
- LOB columns with unlimited size are not supported in the CHANGE_RECORD and BU_CHANGE_RECORD columns. The other fields will be recorded but the LOB will have a NULL value.
- For a description of the audit table structure, see the Qlik Replicate Setup and User Guide.

- **Audit table schema:** Specify a schema if you do not want the Audit table to be created under the target endpoint's default schema.

The default schema are as follows:

Default schema by endpoint

Endpoint	Default Schema
Amazon Redshift	Public
Oracle	The connected user's user name.
Teradata	The endpoint name.
All others	The user's default schema.

- **Audit table tablespace:** This option is only available when the task's target endpoint is Oracle. Enter the tablespace name on the target where you want the Audit table to be created. If you do not enter any information in this field, then the tables will be created in the default permanent tablespace.
- **Audit table name:** Specify a name for the Audit table. The default value is `attrep__audit_table`.

Audit table creation:

If audit table exists when the target is reloaded: Select one of the following to determine how you want to handle the Audit table when the target is reloaded:

- **DROP and CREATE audit table:** The Audit table is dropped and a new table is created in its place.
- **ARCHIVE and CREATE audit table:** A copy of the existing table will be saved to the same schema before the new table is created. The archived table name will be appended with a timestamp, indicating when the archiving operation occurred (e.g. `attrep_audit_table_20170605175601`).



Currently this option is only available for the Hadoop target endpoint.

- **Delete old changes and store new changes in existing audit table:** Data is truncated and added without affecting the Audit table metadata.
- **Keep old changes and store new changes in existing audit table:** Data and metadata of the existing Audit table are not affected.

For a description of the audit table structure, see the Qlik Replicate Setup and User Guide.

Change Processing Tuning

Click the **Change Processing Tuning** sub-tab to fine-tune the Apply Changes settings.

Change Processing Mode

Determine which method will be used to apply changes.



*Changes to tables without a Unique Index or Primary Key will always be applied in **Transactional apply** mode.*

- **Transactional apply:** Select this to apply each transaction individually, in the order it is committed. In this case, strict referential integrity is ensured for all tables.



Applying cached events in transactional mode to endpoints that do not enforce constraints (such as Netezza Performance Server), may result in duplicate records on the target. This is because such endpoints do not return duplicate errors.

- **Batch optimized apply:** Select this to commit the changes in batches. In this case, a pre-processing action occurs to group the transactions into batches in the most efficient way. This may affect transactional integrity. Therefore, you must select one of the following to determine how the system will handle referential integrity issues:

 In the event of a recoverable error during the Batch optimized apply process, multiple `attrep_changes` (Net Changes) tables might be created in the target database (each with a unique name). These tables will need to be deleted manually as the automatic cleanup process will not delete them.

- Preserve transactional integrity

 This option is only supported when replicating to an Oracle target.

- Allow temporary lapses in transactional integrity to improve performance

 The following target endpoints do not support applying binary data types in Batch optimized apply mode:

ODBC, SAP Sybase ASE, Netezza Performance Server, Teradata, and Amazon Redshift.

When LOB columns are included in the replication, **Batch optimized apply** can only be used with the **Limit LOB size to** option. For more information about including LOB columns in the replication, see [Metadata \(page 180\)](#).

Using Batch optimized apply to apply changes to tables with foreign keys is not supported.

Batch tuning

The following options are available when **Batch optimized apply** is selected as the **Change Processing Mode**:

- **Apply batched changes in intervals:**
 - **Longer than:** The minimum amount of time to wait between each application of batch changes. The default value is **1**.
Increasing the **Longer than** value decreases the frequency with which changes are applied to the target while increasing the size of the batches. This can improve performance when applying changes to target endpoints that are optimized for processing large batches, such as Teradata.
 - **But less than:** The maximum amount of time to wait between each application of batch changes (before declaring a timeout). In other words, the maximum acceptable latency. The default value is **30**. This value determines the maximum amount of time to wait before applying the changes, after the **Longer than** value has been reached.
- **Force apply a batch when processing memory exceeds (MB):** The maximum amount of memory to use for pre-processing in **Batch optimized apply mode**. The default value is **500**.
For maximum batch size, set this value to the highest amount of memory you can allocate to Qlik Replicate. This can improve performance when applying changes to target endpoints that are optimized for processing large batches, such as Teradata.
- **Apply batched changes to multiple tables concurrently:** Selecting this option should improve performance when applying changes from multiple source tables.

- **Maximum number of tables:** The maximum number of tables to apply batched changes to concurrently. The default is five, the maximum is 50, and the minimum is two.



Limitations:

When the **Apply batched changes to multiple tables concurrently** option is enabled, the following limitations apply:

- Supported by the following target endpoints only: Snowflake on Azure, Snowflake on AWS, Snowflake on Google, Microsoft SQL Server, Amazon Redshift, Microsoft Azure Databricks Delta, Microsoft Azure Synapse Analytics, and Microsoft Azure SQL Database.
- Error handling limitations:
The global error handling policy will be unavailable.
The task error handling policy defaults will be unchanged for **Environmental** and **Table** errors, but the defaults for **Data** errors and **Apply Conflicts** errors will be as follows:

Data Errors:

- **Data truncation errors:** Ignore record
- **Other data errors:** Suspend table

Apply Conflicts Errors:

- **Deletes:** Ignore record
- **Inserts:** UPDATE the existing target record
Note that this is not relevant for a Snowflake target (as Snowflake does not support Primary Keys).
- **Updates:** Ignore record

Escalation Action:

- The **Escalation action** for both **Data** errors and **Apply Conflicts** is not supported.
- Control Table limitations:
 - The **attrep_apply_exception** Control Table is not supported. For information on this table, refer to the Qlik Replicate Setup and User Guide.

- **Limit the number of changes applied per change processing statement to:** To limit the number of changes applied in a single change processing statement, select this check box and then optionally change the default value. The default value is **10,000**.



This option is not available for tasks configured with the following target endpoints:

- Microsoft SQL Server
- Microsoft Azure SQL Database
- Databricks Lakehouse (Delta) (When using Microsoft Azure Data Lake Storage (ADLS) Gen2 storage)
- Microsoft Azure Synapse Analytics
- Google Cloud SQL for SQL Server
- Amazon Redshift
- Snowflake on AWS
- Snowflake on Azure
- Snowflake on Google

The following options are available when **Transactional apply** is selected as the **Change Processing Mode**:

- **Retrieve all source columns on UPDATE:** To perform INSERTs on the target when UPDATEs are either not possible (for example, due to a missing target record), or because the associated replication task is configured to use Batch Optimized Apply mode (in which case, DELETE + INSERT operations are performed), Replicate needs to retrieve all of the source table columns. This requires full logging (sometimes referred to as “supplemental logging”) to be enabled on all of the source table columns.



- This option is only available for Log Stream Staging tasks.
- This option is only available for data sources that support full logging.
- This option will be selected by default if Replicate detects that full logging is supported in the selected data source. If full logging is supported but not enabled, Replicate will turn on full logging for the selected source tables when the task starts.
- If this option is not enabled, the following task settings should not be selected in the replication task (otherwise, an error will occur during runtime):
 - Error handling > Apply Conflicts: **No record found for applying an UPDATE: INSERT the missing target record**
 - Change Processing > Change Processing Tuning: **DELETE and INSERT when updating a primary key column**

- **Minimum number of changes per transaction:** The minimum number of changes to include in each transaction. The default value is **1000**.



*Replicate applies the changes to the target either when the number of changes is equal to or greater than the **Minimum number of changes per transaction** value OR when the batch timeout value is reached (see below) - whichever occurs first. Because the frequency of changes applied to the target is controlled by these two parameters, changes to the source records may not immediately be reflected in the target records.*

- **Maximum time to batch transactions before applying (seconds):** The maximum time to collect transactions in batches before declaring a timeout. The default value is 1.

Transaction offload tuning

The following tuning options are available, regardless of which **Change processing mode** is selected:

- **Offload transactions in progress to disk if:**
Qlik Replicate usually keeps transaction data in memory until it is fully committed to the source and/or target. However, transactions that are larger than the allocated memory or that are not committed within the specified time limit will be offloaded to disk.
 - **Total transactions memory size exceeds (MB):** The maximum size that all transactions can occupy in memory before being offloaded to disk. The default value is **1000**.
 - **Transaction duration exceeds (seconds):** The maximum time that each transaction can stay in memory before being offloaded to disk. The duration is calculated from the time that Qlik Replicate started capturing the transaction. The default value is **60**.

Miscellaneous tuning

- **Statements cache size (number of statements):** The maximum number of prepared statements to store on the server for later execution (when applying changes to the target). The default is 50. The maximum is 200.
- **Store task recovery data in target database:** Select this option to store task-specific recovery information in the target database. When this option is selected, Replicate creates a table named `atprep_txn_state` in the target database. This table contains transaction data that can be used to recover a task in the event that the files in the **Data** folder are corrupted or if the storage device containing the **Data** folder has failed.
For more information about this option, see [Recovering from data folder loss or corruption \(page 261\)](#).
- **DELETE and INSERT when updating a primary key column:** For tasks configured with streaming target endpoints (for example, Kafka), the DELETE+INSERT statements will be executed on the actual target. For all other target endpoints, they will be executed in the associated Change Tables. This option requires full supplemental logging to be turned on in the source database.
- **Apply changes using SQL MERGE** - When this option is *not* selected, the Batch optimized apply operation executes separate bulk INSERT, UPDATE and DELETE statements for each of the different change types in the Replicate Net Changes table.



The Replicate Net Changes table is created on the target when working in Batch optimized apply mode and contains records that were changed on the source. It is truncated each time the source changes are applied to the target. For more information on the Net Changes table, see [Net Changes table \(page 14\)](#)

While this method is highly efficient, enabling the **Apply changes using SQL MERGE** option is even more efficient when working with endpoints that support this option.

This is due to the following reasons:

- It reduces the number of SQL statements run per table from three to one. Most UPDATE operations in large, immutable, file-based cloud databases (such as Google Cloud BigQuery), involve rewriting of affected files. With such operations, the reduction of per-table SQL statements from three to one is very significant.
- The target database only needs to scan the Replicate net changes table once, significantly reducing I/O.
 - **Optimize inserts:** When **Apply changes using SQL MERGE** is selected together with this option and the changes consist of INSERTs only, Replicate will perform INSERTs instead of using SQL MERGE. Note that while this will usually improve performance and thereby reduce costs, it might also result in duplicate records in the target database.



- The **Apply changes using SQL MERGE** and **Optimize inserts** options are available for tasks configured with the following target endpoints only:
 - Google Cloud BigQuery
 - Databricks Lakehouse (Delta)
 - Snowflake on Azure, Snowflake on Google, and Snowflake on Amazon
- The **Apply changes using SQL MERGE** and **Optimize inserts** options are not supported with the following source endpoints:
 - Salesforce
 - Oracle

For more information, see the "Limitations and Considerations" topic for these endpoints.

- When the **Apply changes using SQL MERGE** option is enabled (either alone or together with the **Optimize inserts** option):
 - Non-fatal data errors or data errors that cannot be recovered will be handled as table errors
 - The [Global error handling policy](#) will not be available
 - The [Apply Conflicts error handling policy](#) will be preset and read-only. If the **Optimize inserts** option is also selected, the **Duplicate key when applying INSERT** Apply Conflicts error-handling option will be set to **Allow duplicates in the target**.
 - Some of the [Data error handling policy](#) options will not be available.
 - The operations will only be performed on the final target tables. For Change Tables or the Audit Table, INSERTs will be performed.

Error handling

Qlik Replicate handles different types of errors during its operation. The way the system should respond to these errors depends on several aspects, including the component where the error occurred, the type of error, and the scope of the error. Because different sites may have different requirements for error behavior, Qlik Replicate lets you configure the error handling.

You can also add an environment variable that instructs Replicate to create dump files in the event of a crash. The dump files can then be used by Qlik Support to troubleshoot the cause of the crash. For more information, refer to the *Qlik Replicate* setup and User Guide.

You can determine whether or not to override the global error handling settings. For more information, see [Error Handling Settings](#).



The option to set a Global Error Handling policy is not available when the **Apply batched changes to multiple tables concurrently** option is selected in the Task Settings' [Change Processing Tuning](#) tab.

Clicking the **Error Handling** tab in the **Task Settings** window reveals the following sub-tabs:

- **Environmental Errors:** An error that is caused by an environmental problem in the source or target endpoint or on the network. Some examples of environmental errors are loss of communication with the source or target endpoint, restarting a database, or network problems.
- **Data Errors:** An error related to data processing at the record level. Some examples of data errors are conversion errors, errors in transformations, or bad data.
- **Table Errors:** An error in processing data or metadata for a specific table. This only includes general table data and not an error that relates to a specific record.
- **Apply Conflicts:** Errors that occur when the target endpoint is not synchronized with the source endpoint when processing changes.
This can cause duplicate key errors on INSERT operations or zero rows affected on UPDATE/DELETE operations.

Error Handling settings

The option to switch between the Global Error Handling policy and a Task Error Handling policy is available in each of the **Error Handling** sub-tabs. However, the policy you enable will be applied to all error types, regardless of where it was enabled. For example, you cannot enable a Task Error Handling policy for Data Errors and then enable the Global Error Handling policy for Table Errors and Environmental Errors.

For information on setting the global error handling policy, see the Qlik Replicate Setup and User Guide.

To set a Task-Specific Error Handling policy:

- Click the **Change to Task Policy** button in any of the **Error Handling** sub-tabs.

To revert to the Global Error Handling policy:

1. Click the **Change to Global Policy** button in any of the **Error Handling** sub-tabs.
2. Click **OK** when prompted to confirm your action.

Environmental errors

Click the **Environmental Errors** sub-tab and then click [Change to Task Policy](#) to configure the following:

- **Maximum retry count:** Select this option and then specify the maximum number of attempts to retry a task when a recoverable environmental error occurs.
 1. To never retry a task, clear the check box or specify "0".
 2. To retry the task an infinite number of times, specify "-1" (the global error handling default).When the system attempts to retry the task the designated number of times, the task is stopped and manual intervention is required.
- **Interval between retry attempts:** Use the counter to select or type the number of seconds that the system waits between attempts to retry a task.
Valid values are 0-2,000.
- **Increase retry interval for long outages:** Select this check box to increase the retry interval for long outages. When this option is enabled, Replicate doubles the interval between each retry attempt and the next, until the **Maximum retry interval** is reached (and continues retrying according to the specified maximum interval).

- **Maximum retry interval:** Use the counter to select or type the number of seconds to wait between attempts to retry a task when the **Increase retry interval for long outages** option is enabled. Valid values are 0-2,000.

Data Errors

Click the **Data Errors** sub-tab and then click [Change to Task Policy](#).



Data error handling is not relevant for Full Load replication. It is applicable to Apply Changes (CDC) replication only.

Data truncation errors

For data truncation errors: When a truncation error occurs in one or more records, Replicate will do one of the following, according to your selected action:

- **Ignore record:** The task continues and the error is ignored.
- **Log record to the exceptions table** (default): The task continues and the error is written to the exceptions table.
- **Suspend table:** The task continues, but data from the table with the error record is moved into an error state and its data is not replicated
- **Stop task:** The task is stopped and manual intervention is required.



Data truncation error handling is supported with the following target endpoints only:

- *MySQL*
- *Google Cloud SQL for MySQL*
- *PostgreSQL*
- *Google Cloud SQL for PostgreSQL*
- *Google Cloud AlloyDB for PostgreSQL*
- *Microsoft Azure SQL Database*
- *Microsoft SQL Server*
- *Google Cloud SQL for SQL Server*
- *Google Cloud BigQuery*
- *Microsoft Fabric Data Warehouse*
- *Oracle*
- *SAP Sybase ASE*
- *File*
- *Amazon Redshift*
- *Databricks Lakehouse (Delta)*
- *Databricks (Cloud Storage)*

NOT NULL constraint violations

For source tables, Replicate can explicitly check for NULL values in each data column of each record, and whether the same column in the corresponding target table is marked as NOT NULL. In such cases, applying the record to the target will trigger a NOT NULL constraint violation that will be difficult to recover from if the task is running in Batch Optimized Apply mode. Therefore, the record is not applied to the target. Instead, the record is handled as an error (see below) without affecting other records in the batch being applied. While this way of handling NOT NULL constraint violation is very effective, it can impact performance, and if the target database does not actually enforce NOT NULL constraint violations, Replicate can be configured to skip that check in order to save CPU time and improve performance. Because Replicate generally knows what target databases enforce NOT NULL constraint violations, it is safe to always let Replicate activate this protection based on the target endpoint type. Still, Replicate gives you the option to override this default and either force a check (with its slight overhead) or disable the check and risk NOT NULL constraint violations.



When a batch update fails due to a NOT NULL constraint violation, Replicate has no way of knowing which of the records in the batch caused that violation. Consequently, Replicate switches to a "one-by-one" strategy of applying changes. While this strategy is acceptable for OLTP databases (although even with those databases, it is much slower than Batch optimized apply), it is not practical for analytical data warehouses, specifically those that use columnar storage in files or micro-partitions (for example, Snowflake, Azure Synapse Analytics, Google BigQuery, and Databricks). As such databases are not designed for one-by-one record updates, performing such updates might take an unreasonable amount of time and consume excessive resources. Moreover, if the target database is a cloud data warehouse that charges based on activity, the one-by-one error handling strategy could result in extremely high costs.

Supported target endpoints

Handling of NOT NULL constraint violations is supported with the following target endpoints:

- All Snowflake-based targets
- Microsoft Fabric Data warehouse
- Amazon Redshift
- Google Cloud BigQuery
- Microsoft Azure Synapse Analytics

Available settings

For NOT NULL constraint violations, select one of the following:

- **Endpoint-determined:** This means that the endpoint type (which can be an OLTP database or an analytical data warehouse) determines whether Replicate checks for NOT NULL constraint violations.



*Some target endpoints do not support checking for NOT NULL constraint violations. In such cases, the tooltip next to the **Endpoint-determined** option will indicate that the current target endpoint does not allow checking for NOT NULL constraints.*

For OLTP-based target endpoints, the default is not to check for NOT NULL constraint violations as it is less critical (from a cost perspective) if a NOT NULL constraint violation occurs during the task. However, if a NOT NULL constraint violation occurs on any of the tables during the task, Replicate will switch to one-by-one mode, which will impact performance to a certain degree (depending on the number and size of the tables being replicated).

You can override the endpoint-determined handling by selecting either the **Check** or **Don't check** options described below.



*The string (**Check**) or (**Don't check**) will appear after **Endpoint-determined** according to the selected target endpoint type.*

- **Check:** Select this if you always want Replicate to check for NOT NULL constraint violations. This might be useful if you know beforehand that some of the source table columns contain NULL values and the corresponding target table columns are configured with NOT NULL constraints. In this case, Replicate will not switch to one-by-one mode, but will continue change processing in Batch Optimized Apply mode. If a source table that will violate a NOT NULL constraint is detected during the check, Replicate will take the action described below. The default action is to suspend the table.
- **Don't check:** Select this if you never want Replicate to check for NOT NULL constraint violations. This is recommended if you are confident that no such violations will occur during the task or if you do not mind Replicate switching to one-by-one mode if a NOT NULL constraint violation occurs.

Handling options when a NOT NULL constraint violation is discovered during a check:

For the **Endpoint-determined** and **Check** options, you can select how Replicate will handle the NOT NULL constraint violation:

- **Suspend table:** This is the default. The task continues, but tables with NULL values are suspended and their data is not replicated.
- **Ignore record:** The task continues and the error is ignored.
- **Log record to the exceptions table:** The task continues, but records with NULL values are written to the exceptions table.
- **Stop task:** The task is stopped and manual intervention is required.

Other data errors

For other data errors: When a data error occurs in one or more records, Replicate will do one of the following, according to your selected action:

- **Ignore record:** The task continues and the error is ignored.
- **Log record to the exceptions table (default):** The task continues and the error is written to the exceptions table.
- **Suspend table:** The task continues, but data from the table with the error record is moved into an error state and its data is not replicated.
- **Stop task:** The task is stopped and manual intervention is required.

Escalating error handling

- **Escalate error handling when other data errors reach (per table):** Select this check box to escalate error handling when the number of non-truncation data errors (per table) reaches the specified amount. Valid values are 1-10,000.



The escalation options are not available when the [Apply changes using SQL MERGE](#) task setting is enabled.

- **Escalation action:** Choose what action Replicate should perform when error handling is escalated. Note that the available actions are dependent on the action selected from the **For other data errors** drop-down list described above.
 - **Log record to the exceptions table:** The task continues, but the record with the error is written to the exceptions table.
 - **Suspend table (default):** The task continues, but data from the table with the error record is moved into an error state and its data is not replicated.



The behavior differs according to the Change Processing Mode:

- *In **Transactional apply** mode, the last changes will not be replicated*
- *In **Batch optimized apply** mode, a situation is possible where there will be no replication of data or data replication will occur in part*

- **Stop task:** The task is stopped and manual intervention is required.

Table Errors

Click the **Table Errors** sub-tab and then click [Change to Task Policy](#) to configure the following:

When encountering a table error: Select one of the following from the drop-down list:

- **Suspend table (default):** The task continues but data from the table with the error record is moved into an error state and its data is not replicated
- **Stop task:** The task is stopped and manual intervention is required.

Escalate error handling when table errors reach (per table): Select this check box to escalate error handling when the number of table errors (per table) reaches the specified amount. Valid values are 1-10,000.

- **Escalation action:** The escalation policy for table errors is set to **Stop task** and cannot be changed.

Apply Conflicts



The Apply Conflicts error handling policy will be preset and uneditable when the [Apply changes using SQL MERGE](#) task setting is enabled.

Click the **Apply Conflicts** sub-tab and then click [Change to Task Policy](#) to configure the following:

No record found for applying a DELETE: Click the triangle to open the list and select what happens when there is a conflict with a DELETE operation. You can select one of the following from the list:

- **Ignore record** (default): The task continues and the error is ignored.
- **Log record to the exceptions table:** The task continues and the record is written to the exceptions table.
- **Suspend table:** The task continues but data from the table with the error record is moved into an error state and its data is not replicated.
- **Stop task:** The task is stopped and manual intervention is required.

Duplicate key when applying an INSERT: Click the triangle to open the list and select what happens when there is a conflict with an INSERT operation. You can select one of the following from the list:

- **Ignore record:** The task continues and the error is ignored.
- **Log record to the exceptions table** (default): The task continues and the record is written to the exceptions table.
- **Suspend table:** The task continues but data from the table with the error record is moved into an error state and its data is not replicated.
- **Stop task:** The task is stopped and manual intervention is required.
- **Update the existing target record:** The target record with the same primary key as the INSERTED source record is updated.
- **Allow duplicates in the target:** Allows duplicate primary key records to be added to the target table. When **Optimize inserts** is enabled, this is the only option that is available at task level error handling.
For more information on the **Optimize inserts** option, see [Change Processing Tuning \(page 197\)](#).

No record found for applying an UPDATE: Click the triangle to open the list and select what happens when there is a conflict with an UPDATE operation. You can select one of the following from the list:

- **Ignore record:** The task continues and the error is ignored.
- **Log record to the exceptions table** (default): The task continues and the record is written to the exceptions table.
- **Suspend table:** The task continues but data from the table with the error record is moved into an error state and its data is not replicated.
- **Stop task:** The task is stopped and manual intervention is required.
- **Insert the missing target record:** The missing target record will be inserted into the target table. When the source endpoint is Oracle, selecting this option requires supplemental logging to be enabled for all the source table columns.

Escalate handling when apply conflicts reach (per table): Select this check box to escalate error handling when the number of apply conflicts (per table) reaches the specified amount. Valid values are 1-10,000.



- When this option is selected and the number of conflicts reaches the specified number,
- Apply Conflicts errors are counted for each table separately when a task is run. When a task stops, the error count reverts to zero. When a task is resumed/reloaded, the number of errors for all the tables are reset.
- When working in [Batch optimized apply](#) Change Processing mode, the calculation of the Apply Conflicts amount does not include DELETE and UPDATE conflicts that were ignored (as a result of enabling the **Ignore Record** option described above).

Escalation action: Choose what action Replicate should perform when handling is escalated. Note that the available actions are dependent on the action selected in the [drop-down lists described above](#).



When this option is selected and the number of conflicts reaches the specified number, only the escalation action will be performed (i.e. the original action set for the Apply conflict will not be performed).

- **Log record to the exceptions table** (default): The task continues and the error is written to the task log and to the exceptions table.
- **Suspend table:** The task continues but data from the table with the error record is moved into an error state and its data is not replicated.
- **Stop task:** The task is stopped and manual intervention is required.

Logging

You can set the logging level for task logs by selecting the **Logging** tab in the **Task Settings** dialog box and then selecting the **Logging Level** sub-tab. The level you set determines what information is written to the log.



You can also set the task logging level from the **Tools** menu in Monitor view.

For more information, see [Monitor mode \(page 227\)](#) and [Logging \(page 263\)](#).

For more information on monitoring, see [Monitor mode \(page 227\)](#). For more information on setting the task logging level in Qlik Replicate, see the *Qlik Replicate Setup and User Guide*.

The following are the available logging levels. The list is in order from the lowest level to the highest level.

1. Error
2. Warning
3. Info
4. Trace
5. Verbose

The higher levels always include the messages from the lower levels. Therefore, if you select **Error**, only error messages are written to the log. However, if you select **Info**, informational messages, warnings, and error messages are included. Selecting **Verbose** writes all possible messages to the log.

For information on how to set the logging level in Qlik Replicate, see the Qlik Replicate Setup and User Guide.

Storing trace and verbose logging in memory



This option is relevant for Replicate tasks only and is not supported with the following source endpoints:

- *MongoDB*
- *Salesforce (Streaming CDC)*
- *Salesforce (Incremental Load)*
- *SAP Application*
- *SAP ODP*

When the logging level is set to "Trace" or "Verbose", you can instruct Replicate to store the logging information in memory until an error occurs. On detecting an error, Replicate will begin writing to the physical logs and continue to do so for a few minutes after the initial occurrence of the error.

If *no* error occurs before the allocated memory is used up, Replicate will empty the memory buffer and start afresh.

This option is useful for tasks that fail unpredictably and for no obvious reason. The problem with continually writing large amounts of information to the logs is twofold:

- Running in "Trace" or "Verbose" logging mode will quickly use up available disk space (unless the logging settings have been configured to prevent this).
- Continually writing large amounts of data to the logs will affect performance.

To use this option

1. Select the **Store trace/verbose logging in memory, but if an error occurs, write to the logs** check box at the top of the tab.
2. In the **Allocate memory up to (MB)** field, specify the amount of memory you want to allocate for storing logging information.

Character substitution

You can substitute or delete source characters in the target database and/or you can substitute or delete source characters that are not supported by a selected character set.



- All characters must be specified as Unicode code points.
- Character substitution will also be performed on Replicate Control Tables.
- Invalid values will be indicated by a red triangle in the top right of the table cell. Hovering your mouse cursor over the triangle will show the error message.
- Any table-level or global transformations defined for the task will be performed after the character substitution has been completed.
- Substitutions actions defined in the **Substitute or Delete Source Characters** table are performed before the substitution action defined in the **Substitute or Delete Source Characters Unsupported by the Selected Character Set** table.
- Character substitution does not support LOB data types.

Substituting or deleting Source Characters

Use the **Substitute or Delete Source Characters** table to define replacements for specific source characters. This may be useful, for example, when the Unicode representation of a character is different on the source and target platforms. For example, on Linux, the minus character in the `shift_JIS` character set is represented as `U+2212`, but on Windows it is represented as `U+FF0D`.

Substitution actions

To	Do This
Define substitution actions.	<ol style="list-style-type: none"> 1. Click the Add Character button above the table. 2. Specify a source character and a target character in the Source Character and Substitute Character fields respectively. For example to replace the letter "a" with the letter "e", specify <code>0061</code> and <code>0065</code> respectively. <div data-bbox="619 1294 1390 1435" style="border: 1px solid black; padding: 5px; margin: 10px 0;"> <i>To delete the specified source character, enter 0 in the Substitute Character column.</i> </div> 3. Repeat steps 1-2 to replace or delete additional characters.
Edit the specified source or target character	Click anywhere in the relevant column and change the character as required.
Delete entries from the table	Select the desired entry or entries and click the Delete button.

Substituting or deleting Source Characters unsupported by the selected character set

Use the **Substitute or Delete Source Characters Unsupported by the Selected Character Set** table to define a single replacement character for all characters not supported by the selected character set.

Unsupported character substitution actions

To	Do This
Define or edit a substitution action.	<ol style="list-style-type: none"> 1. Select a character set from the Character Set drop-down list in the table. Any characters not supported by the selected character set will be replaced on the target by the character specified in step 2 below. 2. In the Substitute Character column, click anywhere in the column and specify the replacement character. For example, to replace all unsupported characters with the letter "a", enter 0061. <div style="border: 1px solid #ccc; padding: 5px; margin-top: 10px;">  <i>To delete all unsupported characters, enter 0.</i> </div>
Disable the substitution action.	Select the blank entry from the Character Set drop-down list.

File uploads



*The **File Uploads** tab will only be shown if the task is defined with an endpoint that supports this feature.*

Click the **Optimize File Uploads** button to improve performance when replicating to file-based targets such as Amazon S3 and Hadoop. When this feature is enabled, the button label changes to **Disable File Upload Optimization**. Click the **Disable File Upload Optimization** button to disable file upload optimization.

The upload mode depends on the task type:

- **Full Load** - Multiple files created from the same table are transferred in parallel, in no particular order.
- **Apply Changes** - Files created from multiple tables are transferred in parallel. Files created from the same table are transferred sequentially according to creation time.
- **Change Data Partitioning** - Files created from multiple tables and files created from the same table are transferred in parallel.

Note that disabling this option after the task has already started will require you to do one of the following:

- If the task is in the Full Load stage, reload the target using the **Reload Target Run** option.
- If the task is in the Change Processing stage, resume the task using the **Start processing changes from Run** option.

Limitation and considerations

Supported target endpoints

- Supported by the following target endpoints only:
 - Amazon S3
 - Hadoop (Hortonworks and Cloudera)

- Microsoft Azure ADLS
- Databricks (Cloud Storage)
- Microsoft Azure HDInsight
- Hortonworks Data Platform (HDP)
- Google Cloud Storage
- Google Cloud Dataproc
- Amazon EMR
- Cloudera Data Platform (CDP) Private Cloud

General limitations and considerations

- **Post Upload Processing** endpoint settings are not supported.
- **Optimize File Uploads** cannot be enabled together with [Speed partition mode](#).

Hadoop limitations and considerations

- When replicating to a Hadoop target, only **Text** and **Sequence** file formats are supported.
- Hive jobs are not supported as they will prevent the file upload.
- Append is not supported when using **Text** file format.

Amazon S3 and Microsoft Azure ADLS limitations and considerations

- When working with Reference Files, a new entry is added to the Reference File immediately after the data file is uploaded (even if the DFM file has not been uploaded yet).
- The existence of the DFM file does not necessarily mean that the associated data file has also been uploaded.

Message format



*The **Message Format** tab will only be shown if the task is defined with a streaming target endpoint that supports this feature.*

When a task is defined with such an endpoint, you can specify a custom message format that will override the default Replicate message format. This may be useful if the consumer application needs to process the message in a particular format.

The custom message format can be defined at task level and/or at table level. When it is defined at both task *and* table level, the message format defined for the table will take precedence over the message format defined for the task.

For information on defining a custom message format at table level, see [Message format \(page 118\)](#).



- *Supported by the Kafka target endpoint only*
- *The custom message format feature cannot be used in conjunction the Attunity Avro Message Decoder SDK*
- *Hierarchical structured messages are not supported.*

General rules and usage guidelines

When defining a custom message, it is important to consider the rules and usage guidelines listed below.

Section names:

The following naming rules apply:

- Section names must start with the characters a-z, A-Z, or _ (an underscore) and can then be followed by any of the following characters: a-z, A-Z, 0-9, _
- With the exception of the **Record name** and **Key name sections** (that do not end with a slash), removing the slash from section names will flatten the hierarchy of the associated section (see [Slashes](#) below).
- All section names except **Record name** and **Key name** can be deleted (see [Deletion: \(page 216\)](#) below)
- The **Data name** and **Include record Before-data** section names cannot both be deleted
- The **Data name** and **Include record Before-data** section names cannot be the same

Slashes:

Some of the section names in the UI end with a slash (e.g. beforeData/). The purpose of the slash is to maintain a hierarchy of the different sections within the message. If the slash is removed, the following will occur:

- The hierarchical structure of that section will be flattened, resulting in the section name being removed from the message
- The section name will be prefixed to the actual metadata, either directly or using a separator character (e.g. an underscore) that you appended to the name

Example of a data message when headers/ is specified with a slash:

```
"message":{
  "data":{
    "COL1": "159",
    "COL2": "159"
  },
  "beforeData": null,
  "headers": {
    "operation": "INSERT",
    "changeSequence": "2018100811484900000000233",
```

Example of a data message when headers_ is specified with an underscore instead of a slash:

```
"message":{
  "data":{
    "COL1": "159",
    "COL2": "159"
  },
  "beforeData": null,
```

```
"headers_operation": "INSERT",
"headers_changeSequence": "2018100811484900000000233",
```

Deletion:

Deleting a section name from the message will flatten the hierarchical structure of that section. This will result in all of that section's data appearing immediately below the content of the preceding section.

Example of a data message with the `headers` section name:

```
"message":{
  "data":{
    "COL1": "159",
    "COL2": "159"
  },
  "headers": {
    "operation": "INSERT",
    "changeSequence": "2018100811484900000000233",
```

Example of a data message without the `headers` section name:

```
"message":{
  "data":{
    "COL1": "159",
    "COL2": "159"
  },
  "beforeData": null,
  "operation": "INSERT",
  "changeSequence": "2018100811484900000000233",
```

Variables

You can add variables to names by clicking the *fx* button at the end of the row. The following variables are available:

- `SERVER_NAME` - The host name of the Replicate Server
- `TARGET_TABLE_NAME` - The name of the table
- `TARGET_TABLE_OWNER` - The table owner
- `TASK_NAME` - The name of the Replicate task



The `TARGET_TABLE_OWNER` variable is not available for the **Record name** and **Key name** options (described in the table below).

Defining a custom message format

To define a custom message format, click the **Custom Message Format** button and configure the options as described in the table below.

To revert to the default message format, click the **Default Message Format** button.

Message format options

Option	Description
Include Namespace	<p>A unique identifier for the message. This should be a string, separated by periods. Note that the Namespace will be included in both the message and the message key.</p> <p>Example:</p> <p>mycompany.queue.msg</p> <p>Default name:</p> <p>com.attunity.queue.msg.{{TASK_NAME}}.{{TARGET_TABLE_OWNER}}.{{TARGET_TABLE_NAME}}</p> <p>Default name for Control Tables:</p> <p>com.attunity.queue.msg.{{TARGET_TABLE_NAME}}</p> <div data-bbox="357 835 1390 1794" style="border: 1px solid #ccc; padding: 10px; margin-top: 10px;"> <p> <i>The default Control Table Namespace cannot be changed via the UI. You can, however, change the default Control Table Namespace as follows:</i></p> <ol style="list-style-type: none"> <i>In the Task Settings Message Format tab, click the Custom Message Format button and then click OK.</i> <i>Save the task and then export it using the Export Task toolbar button.</i> <i>Open the exported JSON file and add the <code>control_table_namespace</code> parameter as follows (replacing <code>myNamespace</code> with your own value):</i> <pre data-bbox="564 1173 1592 1563" style="background-color: #f0f0f0; padding: 10px;"> "task_settings": { "source_settings": { }, "target_settings": { "queue_settings": { "use_custom_message": true, "message_shape": { "control_table_namespace": "MyNameSpace" } }, "use_custom_key": true, "key_shape": { </pre> <i>Save the JSON file and then import it to Replicate using the Import Task toolbar button.</i> <p><i>For instructions on exporting/importing tasks, see Exporting and importing tasks (page 83).</i></p> </div>

Option	Description
Record name	<p>The name of the record (i.e. message).</p> <p>Default name:</p> <p>DataRecord</p>
Data name	<p>All of the data columns included in the record.</p> <p>Default name:</p> <p>data/</p>
Include Headers	<p>Header columns provide additional information about the source operations.</p> <p>For more information, see Include the following headers (page 218) below.</p>
Include headers namespace	<p>A unique identifier for the header columns section. This should be a string, separated by periods.</p> <p>Example:</p> <p>headers.queue.msg</p> <p>Default name:</p> <p>com.attunity.queue.msg</p>
Headers name	<p>The name of the section containing the Replicate column headers.</p> <p>Default name:</p> <p>headers/</p>
Include the following headers	<p>For a description of the available header columns, refer to Data Messages in the Qlik Help.</p> <p>Default:</p> <p>All columns are included by default, except the External Schema ID column.</p>
Include record Before-data	<p>When this check box is selected (the default), both pre and post UPDATE data will be included in UPDATE messages.</p> <p>To include only the post UPDATE data in messages, clear the check box.</p> <p>Default name:</p> <p>beforeData/</p>

Option	Description
Key name	<p>The name of the section containing the message key.</p> <p>Default name:</p> <p>keyRecord</p> <p>This option is only applicable when:</p> <ul style="list-style-type: none"> • The Encode the message key in Avro format option is enabled • The Metadata Publishing Method is set to Publish data schemas to Confluent Registry Schema or Publish data schemas to Hortonworks Schema Registry • The Message key is not set to None

Transformations and filters

Click the **Transformations and Filters** tab to show the **Expression defaults** tab. Expressions can be used in global transformations and filter rules as well as in table-level transformations and filters.

This option can be set globally for all tasks or individually for a specific task. To set the option for a specific task, click the **Change to Task Policy** button and set the option as required. If you want to use the global policy set in the server settings' **Transformations and Filters** tab, click the **Change to Global Policy** button.

- **Support special characters in column names used in expressions:** Enable this option if you want to include source column names with special characters in expressions defined for this task. An example of such a column name would be `special#column`.



A condition for enabling this option is that all table and global transformations must use strict SQL syntax. Specifically, all string literals must use single quotes (') rather than double quotes. If any of the transformations on any task on the server uses double quotes (") to quote literals, then you should not enable this option.

Before enabling this option, you must fix any use of double quotes in string literals. For example, a transformation such as:

```
CASE age WHEN age < 18 THEN "adult" ELSE "minor" END)
```

must be rewritten as:

```
CASE age WHEN age < 18 THEN 'adult' ELSE 'minor' END)
```

See also:

- [Defining global rules \(page 119\)](#)
- [Using filters \(page 103\)](#)
- [Defining transformations for a single table/view \(page 93\)](#)
- [Using the Expression Builder \(page 149\)](#)

More options

These options are not exposed in the UI as they are only relevant to specific versions or environments. Consequently, do not set these options unless explicitly instructed to do so by Qlik Support or product documentation.

To set an option, simply copy the option into the **Add feature name** field and click **Add**. Then set the value or enable the option according to the instructions you received.

10 Monitoring and controlling tasks

By default, Qlik Enterprise Manager opens in **Tasks** View. **Tasks** View is where you monitor and control your Replicate and Compose tasks. To switch between **Servers** View and **Tasks** View, click the **Tasks** tab in the top left of the console.



Currently, the following options are not available for Compose tasks:

- *Task drill-down*
- *Deleting tasks*
- *Enabling/disabling scheduling*
- *Generating task instructions*
- *Monitoring of Command tasks*

For information on managing user permissions for all tasks on a server or for a specific task, see [Managing user permissions \(page 307\)](#).

In this section:

- [Monitoring Replicate tasks \(page 221\)](#)
- [Monitoring Compose tasks and workflows \(page 240\)](#)
- [Searching for tasks \(page 248\)](#)
- [Customizing task columns \(page 249\)](#)
- [Grouping tasks \(page 250\)](#)
- [Running a task \(page 252\)](#)
- [Error handling \(page 262\)](#)
- [Using the monitor tools \(page 263\)](#)
- [Scheduling jobs \(page 266\)](#)

10.1 Monitoring Replicate tasks

In this section:

- [Task progress summary \(page 221\)](#)
- [Viewing specific tasks \(page 225\)](#)
- [Monitoring Full Load replication \(page 228\)](#)
- [Monitoring Change Processing replication \(page 233\)](#)

Task progress summary

The **Tasks** View displays a list of tasks in tabular format in the middle pane.

For each task, Enterprise Manager can display the following columns:

General columns

- **State:** The current state of a task. Possible states are described in the table below. You can filter the table by state by selecting or clearing the respective check box in the **States** pane above the table.

Task states

Task State	Icon	Description
Running		The task is running. <div style="border: 1px solid #ccc; padding: 5px;"> <i>A spinning circle icon will be shown when the task is in a "Starting" state.</i></div>
Stopped		The task has not been run yet or when the task has stopped at some point during the replication. <div style="border: 1px solid #ccc; padding: 5px;"> <i>A spinning circle icon will be shown when the task is in a "Stopping" state.</i></div>
Error		The task has stopped due to a fatal error.
Recovering		The task is recovering from a recoverable error.

- **Server:** The name of the server
- **Task:** The name of the task
- **Stage:** The current stage of the task: Loading or Change Processing.
- **Tables with Error:** The number of tables with error.

Not displayed by default:

- **Reason:** An explanation for the current state
- **Description:** A description of the task, as entered by the user
- **Data Errors:** The total number of data errors in all tables involved in the task. The count is affected by data errors and the **Reset Data Errors** option available when you drill down to a task.
- **Profile:** The task profile, which can be either unidirectional or bidirectional
- **Log Stream Staging:** The name of the Log Stream Staging task, both for the actual Log Stream Staging task and for any tasks associated with the Log Stream Staging task. For more information about the Log Stream Staging feature, refer to the *Qlik Replicate Setup and User Guide*.
- **Type:** The type of task, such as Full Load, CDC, or Full Load & CDC
- **Loading Ended:** Indicates whether loading has completed. This refers to completion of the Full Load stage in Replicate tasks and completion of the Full Load and/or Change Processing stage in Compose tasks.
- **Source Name:** The name of the source
- **Source Type:** The database type of the source

- **Target Name:** The name of the target
- **Target Type:** The target's database type
- **Memory (MB):** The current utilization of virtual memory, in MB. A task's virtual memory utilization is sampled approximately every 10 seconds. When the task is not running, the value is set to zero (0).
- **Disk Usage (MB):** The current utilization of disk space, in MB. A task's disk utilization is sampled approximately every minute.
- **CPU (%):** The amount of CPU being used by a specific task. The CPU sampling rate is approximately 10 seconds.
- **Tags:** The tags associated with a task

Loading columns

- **Progress (%):** The percentage of loading completed
- **Target Throughput (rec/sec):** The current target throughput, in rec/sec

Not displayed by default:

- **Started:** The date and time the loading started
- **Ended:** The date and time the loading ended
- **Load Duration:** The duration of the load process, in hh:mm:ss
- **Total Tables:** The total number of tables
- **Tables Completed:** The number of completed tables
- **Tables Left:** The number of tables remaining to be loaded
- **Tables Loading:** The number of tables currently loading
- **Tables Queued:** The number of tables in queue
- **Records Completed:** The number of completed records
- **Records Left:** The number of records remaining to be loaded
- **Source Throughput (kbyte/sec):** The current source throughput, in kbyte/sec
- **Source Throughput (rec/sec):** The current source throughput, in rec/sec
- **Table Notes:** Table notes, such as "0 tables failed loading"
- **Target Throughput (kbyte/sec):** The current target throughput, in kbyte/sec
- **Total Records:** The total number of records



*With Compose tasks, canceled tables are reflected in the **Total Tables** counter but not in the **Completed/Loading/Queued/Error** counters. Therefore, when one or more tables is canceled in a task, the sum of these counters will be less than the **Total Tables**.*

Change Processing columns

- **Incoming Changes:** The number of changes currently being processed
- **Applied Changes:** The number of changes applied
- **Apply Throughput (rec/sec):** The apply throughput, in rec/sec
- **Apply Latency:** The apply latency

- **Apply Throughput (kbyte/sec):** The apply throughput, in kbyte/sec
- **Changes for Apply - In Memory (Target):** The number of changes in memory during apply and until target commit
- **Changes for Apply - On Disk (Target):** The number of changes on disk during apply and until target commit
- **Changes In Memory (Source):** The number of changes accumulated in memory until source commit
- **Changes on Disk (Source):** The number of changes accumulated on disk until source commit
- **COMMIT Change Records:** The number of COMMIT change records
- **COMMIT Change Volume:** The number of COMMIT change volume, in MB
- **COMMIT Transactions:** The number of COMMIT transactions
- **DDLs:** The total number of applied DDLs
- **DDLs (%):** The percentage of applied DDLs
- **DELETES:** The total number of DELETES applied
- **DELETES (%):** The percentage of DELETES applied
- **Incoming Transactions:** The number of incoming transactions
- **INSERTs:** The total number of INSERTs applied
- **INSERTs (%):** The total percentage of INSERTs applied
- **ROLLBACK Change Records:** The number of ROLLBACK change records
- **ROLLBACK Change Volume:** The number of ROLLBACK change volume, in MB
- **ROLLBACK Transactions:** The number of ROLLBACK transactions
- **Source Latency:** The current source latency, in hh:mm:ss
- **Source Throughput (kbyte/sec):** The current source throughput, in kbyte/sec
- **Source Throughput (rec/sec):** The current source throughput, in rec/sec
- **Total Latency:** The overall latency, in hh:mm:ss
- **Transactions (Source):** The number of transactions accumulated until source commit
- **Transactions for Apply (Target):** The number of transaction during apply and until target commit
- **UPDATES:** The total number of UPDATES applied
- **UPDATES (%):** The percentage of UPDATES applied

Status summary

The **Status Summary** at the bottom of the window provides a quick overview of the current status of all monitored tasks and servers. It lists the following information:

- **For tasks:** The total number of monitored tasks as well as the number of tasks that are running (green icon), stopped (gray icon), recovering (orange icon), and encountered an error (red icon)



The "running" state also includes "starting" tasks. Similarly, the "stopped" state also includes "stopping" tasks.

You can double-click the task counters to open the corresponding **Tasks** View.

Note: The task counters do not consider tasks on servers that are not currently monitored. For more information, see [Additional server management options \(page 64\)](#).

- **For servers:** The total number of servers, the number of servers that are running and successfully monitored (green icon), the number of faulty servers that Enterprise Manager cannot monitor (red icon), and the number of servers that are not being monitored (gray icon)

Status Bar Example



Viewing specific tasks



Currently, this functionality is available for Replicate tasks only.

From the **Tasks** view, you can drill down to an individual task, provided you have already created at least one task (see [Defining and managing tasks \(page 69\)](#) for more information). Two modes display different sets of information for each task:

- [Designer mode \(page 225\)](#): Here you define endpoints, select tables, modify table settings (including filters and transformations), and create global transformation rules.
- [Monitor mode \(page 227\)](#): Default mode when you open a task. Here you view replication task activities in real time, along with log messages and notifications.

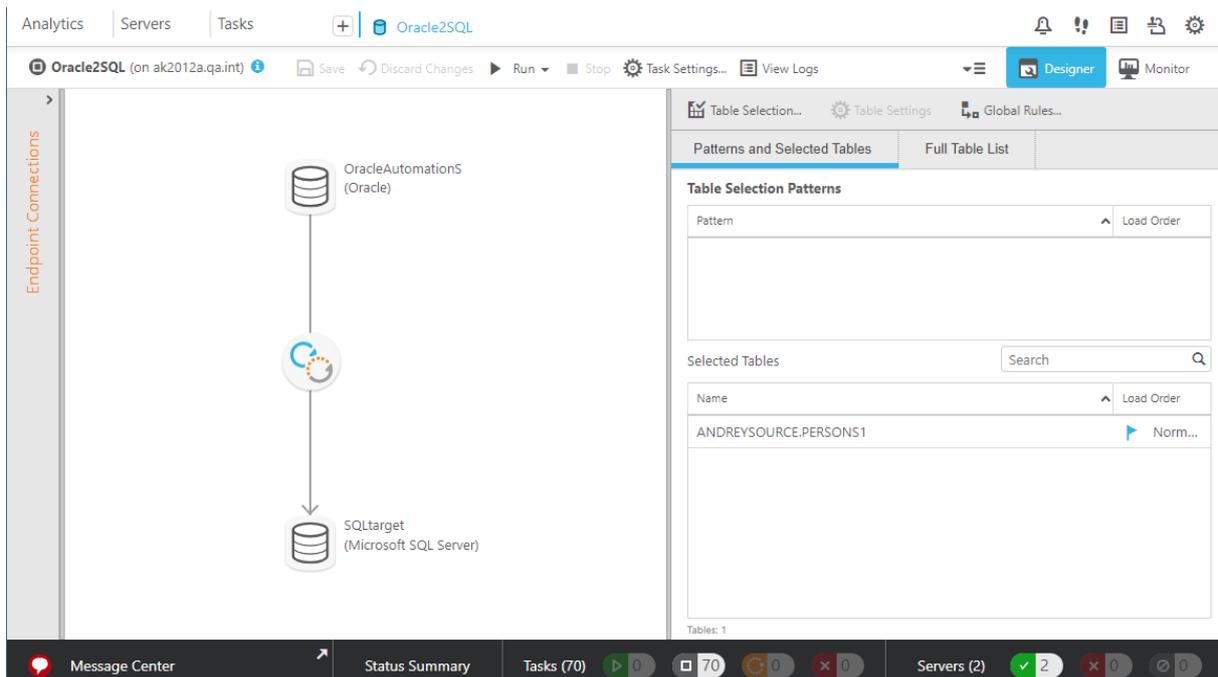
To view a specific task:

1. In the **Tasks** view, select the task you want to work with.
The right pane displays the task diagram on the right side of the page.
2. On the **Tasks** view toolbar, click **Open**.

Designer mode

In **Designer** mode, you define endpoints, select tables to be replicated, modify table settings (including filters and transformations), and create global transformation rules.

Viewing a task in Designer mode



The **Designer** mode includes the following elements:

- **Endpoints list:** Lists the source and target endpoint connections that you added to Qlik Replicate. For more information, see [Defining and managing endpoints \(page 85\)](#). The figure shows the Endpoints List in a collapsed state, hiding the endpoints. To expand the list, click the right arrow at the top or anywhere below it. To close the panel, click the left arrow.
- **Endpoints map:** Illustrates the connection between the source and target endpoints for the task. The round icon between the endpoints represents the task type, which can indicate Full Load only, Full Load and Apply Changes, or Apply Changes only.
- When you create a task, you can drag the endpoints to the source and target drop spots as required. For more information, see [Adding a source and target endpoint to a task \(page 74\)](#).
- **Monitor and Designer buttons:** Lets you switch between **Monitor** mode and **Designer** mode. See also [Monitor mode \(page 227\)](#).
- **Run button:** Lets you run the task at hand.
- **Task Settings button:** Opens the **Task Settings** dialog box. For more information, see [Task Settings \(page 179\)](#).
- **Manage Endpoint Connections button:** Lets you view the endpoints defined, edit them, or add new endpoints. For more information, see [Defining and managing endpoints \(page 85\)](#).
- **Select and Define Tables:** Lets you select the tables you want to include in your replication task. In addition, you can use transformation and filter operations to create new tables or to replicate parts of tables. For more information, see [Selecting tables and/or views for replication \(page 76\)](#), [Using filters \(page 103\)](#), and [Defining transformations for a single table/view \(page 93\)](#).
- **Global Transformations option:** Lets you create transformations for all tables in a task. For more information, see [Starting the Global Transformation Rules wizard \(page 120\)](#).

To display a task in Designer mode:

- On the right side of the toolbar, click **Designer**.

Monitor mode

In **Monitor** mode, you view the replication task activities in real time. This is the default mode when you open a task.

Viewing a task in Monitor mode

The screenshot shows the Enterprise Manager interface for a replication task named 'Task1'. The task is in 'Monitor' mode, as indicated by the 'Monitor' button being active in the top right. The main area displays the task's progress and details. A 'Full Load' progress bar is at 100%. The 'Ended Date' is Sep 1, 2016, 9:48:01 AM, and the 'Load Duration' is 00:01:52. A bar chart shows the status of tables: 7 Completed, 0 Loading, 0 Queued, and 0 Error. Below this, a table list is visible with columns for Table Name, Load Duration, Estimated Count, Transferred Count, Current Through..., and Cached Changes. On the right, a 'Task Map' diagram shows the data flow from the source endpoint 'src (Microsoft SQL Server)' to the target endpoint 'sqltrgt (Microsoft SQL Server)'.

The **Monitor** mode includes the following elements:

- **Run button:** Lets you run the task at hand.
- **Manage Endpoint Connections button:** Lets you view the endpoints defined, edit them, or add new endpoints. For more information, see [Defining and managing endpoints \(page 85\)](#).
- **Monitor and Designer buttons:** Switch between **Monitor** mode and **Designer** mode. See also [Designer mode \(page 225\)](#) and [Defining and managing tasks \(page 69\)](#).
- **Tools list:** Provides access to history, log management, and status information.
- **Change Processing/Full Load tabs:** Lets you select the information you want to focus on. By default, Enterprise Manager displays the **Change Processing** view.
- **Task Map:** Illustrates the connection between the source and target endpoints for the task. The round icon between the endpoints represents the task type, which can indicate Full Load only, Full Load and Apply Changes, or Apply Changes only.
- **Message Center:** Displays notifications and logging messages. By default, the Message Center is minimized to a message tray at the bottom left of the console. For more information, see [Messages and notifications \(page 269\)](#).

To display a task in Monitor mode:

- On the right side of the toolbar, click **Monitor**.

Monitoring Full Load replication

You can view general information on the progress of a Replicate Full Load operation in the **Loading** tab, which is located in the **Task Dashboard** to the right of the Task List. You can also drill down to more detailed information in a dedicated tab for the task by clicking a bar or gauge in the **Task Dashboard**, or by double-clicking the task in the table.

In this section:

- [General information for loading replication \(page 228\)](#)
- [Detailed information for Full Load replication \(page 228\)](#)

General information for loading replication

The **Loading** tab in the right pane shows the following general progress information:

- **Source:** The name and type of the source endpoint.
- **Target:** The name and type of the target endpoint.
- **Loading** and **Change Processing** tabs containing the information described below.
- **Load Duration:** The duration of the loading operation
- **Ended:** When the loading operation completed
- **Overall Progress bar:** The overall progress of the loading operation
- **Table bars:** The status of the tables being loaded, which can be:
 - **Completed** - The number of tables that have been loaded into the target.
 - **Loading** - The number of tables that are currently being loaded into the target.
 - **Queued** - The number of tables that are waiting to be loaded into the target.
 - **Error** - The number of tables that could not be loaded due to an error
See [Messages and notifications \(page 269\)](#) and [Data error handling \(page 263\)](#) for more information about error messages.
- **Tables:** The number of total tables, the number of tables that have completed loading into the target, and the number of tables remaining to be loaded into the target.
- **Records:** The number of total records, records that have completed loading into the target endpoint, and records remaining to be loaded into the target endpoint
- **Throughput gauge:** The current throughput, which is the number/volume of records being retrieved from the source and applied to the target at any given time
For more information, see [Setting the unit of throughput measurement \(page 233\)](#).

Detailed information for Full Load replication



This functionality is currently available for Replicate tasks only.

You can view more detailed information about the Full Load status by double-clicking a status bar or about throughput by double-clicking the throughput gauge. This opens the task in the <task name> tab, in the context of the bar or gauge that you clicked. The information that is displayed is the same information that you would see if you were monitoring the task in Replicate.

In this section:

- [Information about the overall progress \(page 229\)](#)
- [Information about all tables that are being loaded \(page 229\)](#)
- [Information about tables that Replicate has finished loading \(page 231\)](#)
- [Information about tables that are being loaded \(page 231\)](#)
- [Information about tables that are waiting to be loaded \(page 232\)](#)
- [Information about tables with errors \(page 232\)](#)
- [Information about throughput \(page 233\)](#)
- [Setting the unit of throughput measurement \(page 233\)](#)

Information about the overall progress

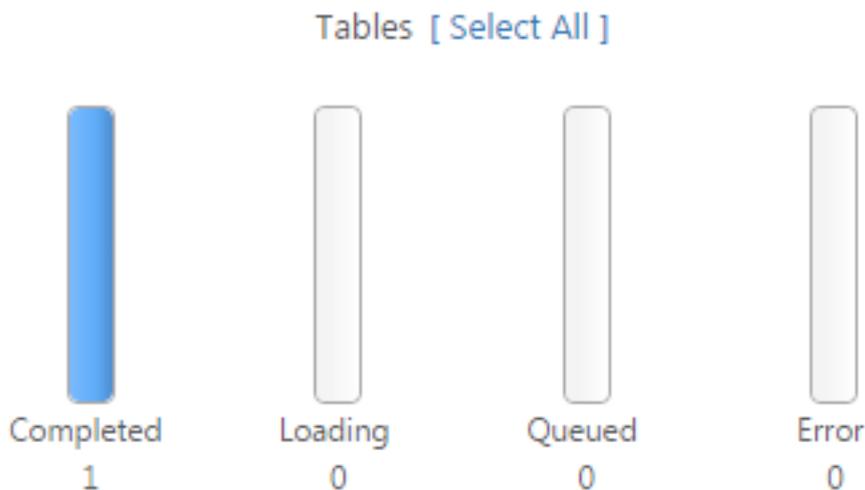
To view additional details about the overall progress, in the <task name> tab, click the **Total Completion** bar. The **Progress Details** area displays a grid with the following information:

Progress information

Category	Total	Completed	Remaining	Notes
Tables	The total number of tables that are included in the task.	The total number of tables that completed loading at the current time.	The total number of tables waiting to be loaded.	Additional information, such as how many tables failed loading.
Records	The total records that are included in the task the current time.	The total number of records that completed loading at the current time.	The total number of records waiting to be loaded.	Additional information.
Time	The estimated time to load all of the selected tables in the task.	The total elapsed time.	The estimated amount of time to load the remaining tables.	Additional information.

Information about all tables that are being loaded

To view additional details about the tables, or to reload data to selected tables, in the <task name> tab, click **Select All**.



The **Tables - All** area displays a grid with the following information:

- **Table Name:** The name of the source table that is included in the task
- **Status:** The task status, which can be any of the following:
 - **Queued** - The table is in the queue waiting to be loaded to the target endpoint.
 - **Loading** - The table is being processed but is not finished loading.
 - **Completed** - All of the table records are loaded to the target.
 - **Error** - The table stopped loading due to an error.
See [Messages and notifications \(page 269\)](#) and [Data error handling \(page 263\)](#) for more information about error messages.



*With Compose tasks, canceled tables are reflected in the **Total Tables** counter but not in the **Completed/Loading/Queued/Error** counters. Therefore, when one or more tables is canceled in a task, the sum of these counters will be less than the **Total Tables**.*

- **Estimated Count:** The estimated number of records that have been loaded to the target
- **Elapsed Time:** The total elapsed time since Replicate started processing the records
- **Progress:** The loading progress in terms of percentage
- **Reload:** To reload selected tables, select the tables you want to reload and then click **Reload** above the table list. When prompted to confirm the operation, click **OK**. The data in the selected tables will be reloaded to the target endpoint. Note that this option is not available for **Apply Changes Only** tasks.

You can also:

- **Reload tables.** To reload selected tables, select the tables you want to reload and then click **Reload** above the table list. When prompted to confirm the operation, click **OK**. The data in the selected tables will be reloaded to the target endpoint. Note that this option is not available for **Apply Changes Only** tasks.

- Use the **Columns Settings** dialog box to select which columns to display and to arrange the order in which they appear. For more information, see [Customizing task columns \(page 249\)](#).
- Hide a column, export the list to a TSV file, or sort by column in ascending or descending order. For more information, see [Customizing task columns \(page 249\)](#).

Information about tables that Replicate has finished loading

To view more information about tables that Replicate has finished loading, or to reload data to selected tables, in the <task name> tab, click the **Completed** bar. The **Tables - Completed** area displays a grid with the following information:

- **Table name:** The name of the source table that has completed loading.
- **Loaded On:** The time that the table completed loading all of its records to the target.
- **Transferred Count:** The number of records loaded to the target.
- **Transferred Volume (MB):** The volume of the records (in KB) loaded to the target.
- **Load Duration:** The amount of time that it took for all records to load to the target.
- **Throughput Records:** The average throughput rate for the table. Throughput describes the number of records read per second. See also [Setting the unit of throughput measurement \(page 233\)](#).
- **Throughput Volume (KB/sec):** The average throughput rate for the table. Throughput describes the volume of records (in KB) read per second. See also [Setting the unit of throughput measurement \(page 233\)](#).

You can also:

- Reload tables. To reload selected tables, select the tables you want to reload and then click **Reload** above the table list. When prompted to confirm the operation, click **OK**. The data in the selected tables will be reloaded to the target endpoint. Note that this option is not available for **Apply Changes Only** tasks.
- Use the **Columns Settings** dialog box to select which columns to display and to arrange the order in which they appear. For more information, see [Customizing task columns \(page 249\)](#).
- Hide a column, export the list to a TSV file, or sort by column in ascending or descending order. For more information, see [Customizing task columns \(page 249\)](#).

Information about tables that are being loaded

To view more information about tables that Replicate is currently loading, or to reload data to selected tables, in the <task name> tab, click the **Loading** bar. The **Tables - Loading** area displays a grid with the following information:

- **Table Name:** The names of the source tables that are currently loading.
- **Load Duration:** The amount of time that it took for all records to load to the current point in time.
- **Estimated Count:** The estimated number of rows that are waiting to be loaded in the full load operation.
- **Transferred Count:** The number of records that are loaded to the target endpoint.
- **Current Throughput:** The current throughput rate for the table. Throughput describes the number of records read per second. For more information about throughput, see also [Setting the unit of throughput measurement \(page 233\)](#).

- **Cached Changes:** The number of changes that were cached during full load when the source tables changed. Cached changes are typically applied to tables after the last table completes full load.
- **Estimated Finish Time:** The approximate date and time the task will finish loading the tables.
- **Progress:** The table status and the time the table entered that status.

You can also:

- Reload tables. To reload selected tables, select the tables you want to reload and then click **Reload** above the table list. When prompted to confirm the operation, click **OK**. The data in the selected tables will be reloaded to the target endpoint. Note that this option is not available for **Apply Changes Only** tasks.
- Use the **Columns Settings** dialog box to select which columns to display and to arrange the order in which they appear. For more information, see [Customizing task columns \(page 249\)](#).
- Hide a column, export the list to a TSV file, or sort by column in ascending or descending order. For more information, see [Customizing task columns \(page 249\)](#).

Information about tables that are waiting to be loaded

To view more information about tables in the loading queue, in the <task name> tab, click the **Queued** bar. The **Tables - Queued** area displays a grid with the following information:

- **Table Name:** The names of the source tables that are currently in the queue waiting to be loaded.
- **Estimated Count:** The estimated number of rows that are waiting to be loaded in the full load operation.

You can also:

- Use the **Columns Settings** dialog box to select which columns to display and to arrange the order in which they appear. For more information, see [Customizing task columns \(page 249\)](#).
- Hide a column, export the list to a TSV file, or sort by column in ascending or descending order. For more information, see [Customizing task columns \(page 249\)](#).

Information about tables with errors

To view more information about tables that could not be loaded due to an error, in the <task name> tab, click the **Error** bar. The **Tables - Error** area displays a grid with the following information:

- **Table Name:** The names of the source tables that could not be loaded to the target.
- **Failed On:** The time that the error occurred.
- **Loaded Count:** The number of records loaded when the error occurred.

You can also:

- Reload tables. To reload selected tables, select the tables you want to reload and then click **Reload** above the table list. When prompted to confirm the operation, click **OK**. The data in the selected tables will be reloaded to the target endpoint. Note that this option is not available for **Apply Changes Only** tasks.

- Use the **Columns Settings** dialog box to select which columns to display and to arrange the order in which they appear. For more information, see [Customizing task columns \(page 249\)](#).
- Hide a column, export the list to a TSV file, or sort by column in ascending or descending order. For more information, see [Customizing task columns \(page 249\)](#).

For more information, see [Data error handling \(page 263\)](#).

Information about throughput

To view a graph representing the number or volume of records (depending on the selected measurement unit) processed during the Full Load operation, click the **Throughput** gauge. The **Throughput Details** area displays a graph illustrating the source and target throughput rate. This rate indicates how fast the table records are being replicated to the target endpoint.

Setting the unit of throughput measurement

You can set the throughput measurement values either to the number of records replicated per second or to the number of kilobytes replicated per second. The display is always based on the current load operation.

To set the unit of throughput measurement:

- From the list above the **Throughput** gauge, select **rec/sec** or **kbyte/sec**.

Monitoring Change Processing replication

You can view general information on the progress of Change Data Capture (CDC) processing in the **Change Processing** tab, which is located in the **Task Dashboard** to the right of the Task List. You also have the option to drill down to more detailed information in a dedicated tab for the task by clicking a bar, graph, or gauge in the **Task Dashboard**, or by double-clicking the task in the table.



This functionality is currently relevant to Replicate tasks only.

In this section:

- [General Change Processing information \(page 233\)](#)
- [Detailed Change Processing information \(page 236\)](#)

General Change Processing information

General information about change processing is presented in a graphical format, as shown in the following figure.

Task Dashboard - Change Processing Status

Full Load
Change Processing

Incoming Changes
0

Transactions
184

Applied Changes

Total
90,000



- Inserts: **36,000** (40%)
- Updates: **18,000** (20%)
- Deletes: **36,000** (40%)
- DDLs: **0** (0%)

Commit	Transactions	Change Records	Change Volume (MB)
	54,036	90,000	0

Rollback	Transactions	Change Records	Change Volume (MB)
	0	0	0

Apply Throughput

rec/sec ▼



Apply Latency

Now



00:00:00

This following information is displayed:

- **Overall Progress bar:** The overall progress of the task.
- **Incoming Changes:** The number of records that were processed for the task.
- **Transactions:** The number of transactions that were processed for the task.
- **Applied Changes:** A pie chart showing information about the processed changes:
 - The number of INSERT operations processed. Roll over the **Inserts** section with your mouse to see the number and percentage of the accumulated inserts.
 - The number of UPDATE operations processed. Roll over the **Updates** section with your mouse to see the number and percentage of the accumulated updates.
 - The number of DELETE operations processed. Roll over the **Deletes** section with your mouse to see the number and percentage of the accumulated deletes.
 - The number of metadata changes (DDL) processed. DDL changes include information about events like changes to table names or to column names.
- **Commit:** The number of committed transactions and change records as well as the change volume, in MB.
- **Rollback:** The number of rolled back transactions and change records as well as the change volume, in MB.
- **Apply Throughput:** A gauge that describes the number of change events read per second. For additional details, see [Information about Change Processing throughput \(page 239\)](#).
- **Apply Latency:** A gauge that displays latency information.
The latency values in Qlik Enterprise Manager measure the time delay (latency) between the time when a change is visible to the source (and committed), and the time when this same change is visible to the target. The display is always based on the current change being applied.

For additional details, see [Information about Change Processing latency \(page 239\)](#).

Detailed Change Processing information

You can view more detailed information about the Change Processing status by double-clicking the status bar, the pie chart, or a gauge, or by double-clicking the task in the table. This opens the task in the <task name> tab, in context of the bar, chart, or gauge that you clicked (if any). The information that displays is the same information that you would see if you were monitoring the task in Replicate.

In this section:

- [Information about incoming changes \(page 236\)](#)
- [Information about applied changes \(page 238\)](#)
- [Information about Change Processing throughput \(page 239\)](#)
- [Information about Change Processing latency \(page 239\)](#)

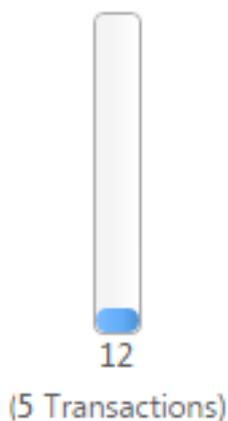
Information about incoming changes

To view more information about incoming changes, in the **Change Processing** tab, double-click the **Incoming Changes** bar. A dedicated tab opens for the task, displaying at the top the **Incoming Changes** bar and at the bottom the **Incoming Changes Details** area with bar graphs that show the following information:

- **Accumulating:** Two bars indicating the number of records currently being read from the source endpoint. These records are accumulated in a queue until they are applied to the target.
 - **In Memory:** The number of accumulating records that are currently in the computer memory.
 - **On Disk:** The number of accumulating records that are currently stored on disk.
- **Applying:** Two bars indicating the number of records currently being written to the target. These are the applied changes.
 - **In Memory:** The number of records being applied that are currently in the computer memory.
 - **On Disk:** The number of records being applied that are currently stored on disk.

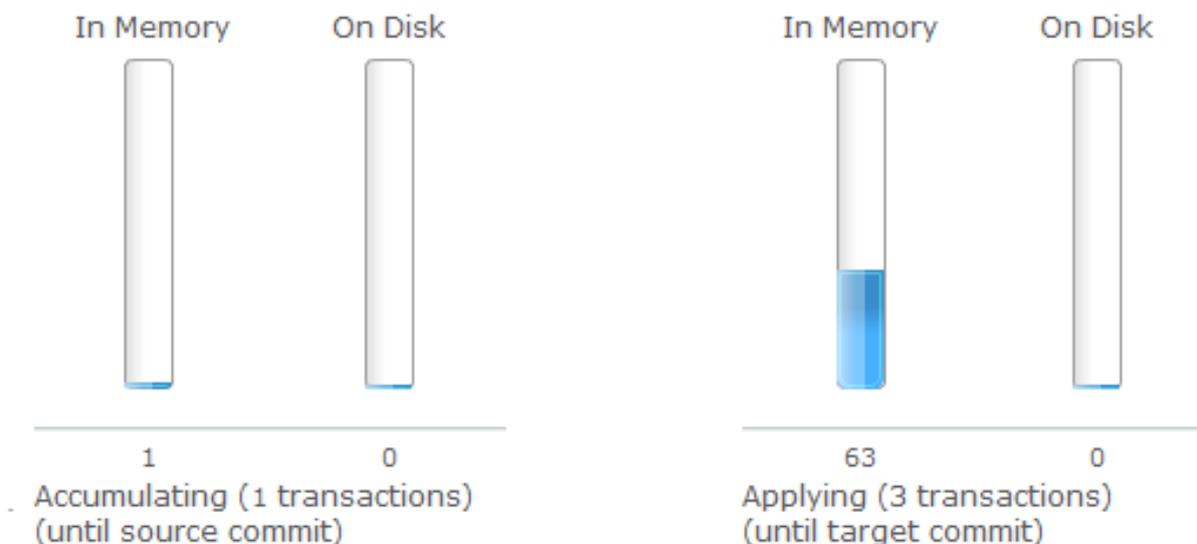
Incoming Changes Bar

Incoming Changes



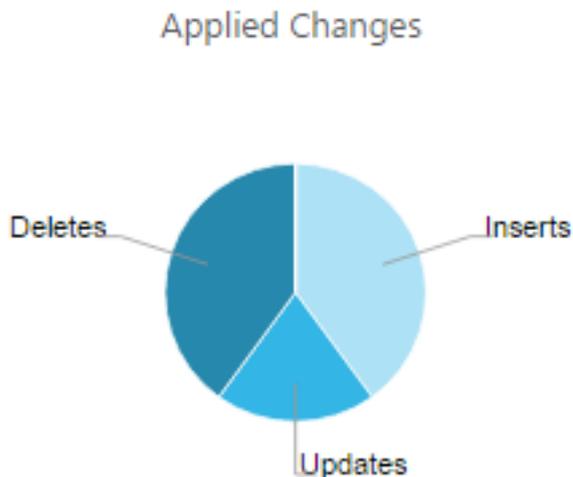
Incoming Changes Details Area

Incoming Changes Details



Information about applied changes

To view more information about applied changes, in the <task name> tab, click the **Applied Changes** pie chart. The **Applied Changes** pie chart displays two grids, each in its own tab, with information about the applied changes.



Recent Activity

The grid in the **Recent Activity** tab includes information about which changes occurred in each table:

- **Table Name:** The names of the source tables that are included in the task.
- **Update:** The number of UPDATE operations processed for the specific table.
- **Insert:** The number of INSERT operations processed for the specific table.
- **Delete:** The number of DELETE operations processed for the specific table.
- **DDL:** The number of metadata changes (DDL) processed. DDL changes include information about events like changes to table names or to column names.
- **Total Applied:** The total number of changes applied to the target.
- **Data Errors:** The number of errors related to data processing at the record level, such as conversion errors, errors in transformations, or bad data.
- **Last Modified:** The time the last change occurred for the specific table.

You can also:

- Filter the grid by entering a string in the **Filter By** box. This allows you to focus on specific tables.
- Reload tables. To reload selected tables, select the tables you want to reload and then click **Reload** above the table list. When prompted to confirm the operation, click **OK**. The data in the selected tables will be reloaded to the target endpoint. Note that this option is not available for **Apply Changes Only** tasks.
- Reset data errors for a specific table. For details, see [Data error handling \(page 263\)](#).
- Use the **Columns Settings** dialog box to select which columns to display and to arrange the order in which they appear. For more information, see [Customizing task columns \(page 249\)](#).

- Hide a column, export the list to a TSV file, or sort by column in ascending or descending order. For more information, see [Customizing task columns \(page 249\)](#).

Aggregates

The grid on the **Aggregates** tab includes information about the total number of changes (for all tables) per change type (INSERT, UPDATE, DELETE, DDL) and transaction type (COMMIT, ROLLBACK).

Information about Change Processing throughput

To view information about how fast change records are loaded to the target endpoint during a change processing operation, in the <task name> tab, click the **Apply Throughput** gauge. The **Apply Throughput Details** area displays a graph illustrating the source and target throughput rate.

See also [Setting the unit of throughput measurement \(page 233\)](#).

Apply Throughput



Information about Change Processing latency

To view information about the time delay (latency) between the time when a change is visible to the source (and committed) and the time when this same change is visible to the target, in the <task name> tab, click the **Apply Latency** gauge. The **Apply Latency Details** area displays a graph illustrating the source and target latency. The display is always based on the current change being applied.

You should take the following into consideration:

- Latency when applying large transactions:
For example, when the most recent latency value was 10 seconds and now a transaction of one million rows gets committed at the source endpoint, Qlik Enterprise Manager starts to apply that transaction to the selected target and it will take some time to write all the changes to the target (for example 60 seconds). During the next 60 seconds, the latency value gradually grows to 70 seconds for the last change in the transaction. Once the transaction is committed, the latency drops back to the 'regular' latency (10 seconds in this case).
- Latency when no transactions are being applied:

When a time period passes with no changes applied to the target, the latency calculation is based on the time difference between the current time and the timestamp of the last change event read from the transaction log. This could happen if, for example, there is high activity on tables that are not selected for replication in the current task.

Apply Latency



10.2 Monitoring Compose tasks and workflows

This section explains how to monitor Compose tasks and workflows in Enterprise Manager.

In this section:

- [Task progress summary \(page 240\)](#)
- [Monitoring Data Lake tasks \(page 243\)](#)
- [Monitoring Data Warehouse tasks \(page 244\)](#)
- [Monitoring workflows \(page 245\)](#)

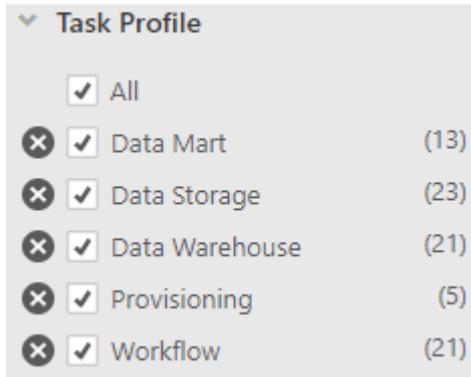
Task progress summary



To see Compose tasks, you need to start the task directly in Compose.

Switch to **Tasks View** to see a list of tasks. Expand the left pane and select the Compose **Task Profile** options according to the tasks or processes that you want to monitor.

Compose task profiles



Data Warehouse tasks:

- Data Warehouse
- Data Mart

Data Lake tasks:

- Data Storage

Compose Workflows:

- Workflows

For each task, the following columns are available.

General columns

- **State:** The current state of a task. Possible states are described in the table below. You can filter the task list using the **States** check boxes above the list.

Task states

Task State	Icon	Description
Running		The task is running. <div style="border: 1px solid #ccc; padding: 5px;"> <i>A spinning circle icon will be shown when the task is in a "Starting" state.</i></div>
Stopped		The task has not been run yet, the task was stopped, or the task completed successfully.
Starting		The task is starting (indicated by a spinning circle)
Error		The task has stopped due to a fatal error.
Recovering		The task is recovering from a recoverable error.

- **Server:** The name of the server on which the task is running.
- **Task:** The name of the task

- **Stage:** The current stage of the task: **Loading** or **Change Processing**.



The stage will remain "Loading", regardless of whether the task is actually loading data or not.

- **Tables with Error:** The number of tables in error state.

Not displayed by default:

- **Description:** A description of the task, as entered by the user in Compose.
- **Loading Ended:** Indicates whether loading has completed. This refers to completion of the Full Load stage in Replicate tasks and completion of the Full Load and/or Change Processing stage in Compose tasks.
- **Project** - The name of the Compose project.
- **Source Name:** The display name of the source database
- **Server type:** Compose
- **Source Type:** The source database type (e.g. Microsoft SQL Server)
- **Target Name:** The display name of the target database
- **Target Type:** The target database type
- **Tags:** The tags associated with a task
- **Type:** The type of task, which can be: **Full Load only**, **CDC only**, or **Full Load & CDC**

Loading columns

- **Progress (%):** The percentage of loading completed

Not displayed by default:

- **Started:** The date and time the loading started
- **Ended:** The date and time the loading ended
- **Load Duration:** The duration of the load process, in hh:mm:ss
- **Total Tables:** The total number of tables
- **Tables Completed:** The number of completed tables
- **Tables Left:** The number of tables remaining to be loaded
- **Tables Loading:** The number of tables currently loading
- **Tables Queued:** The number of tables in queue
- **Total Commands:** The number of commands executed in a Compose Task
- **Commands Completed:** The number of commands completed in a Compose Task



*With Compose tasks, canceled tables are reflected in the **Total Tables** counter but not in the **Completed/Loading/Queued/Error** counters. Therefore, when one or more tables is canceled in a task, the sum of these counters will be less than the **Total Tables**.*

Status summary

The **Status Summary** at the bottom of the window provides a quick overview of the current status of all monitored tasks and servers. It lists the following information:

- **For tasks:** The total number of monitored tasks as well as the number of tasks that are running (green icon), stopped (gray icon), recovering (orange icon), and in error state (red icon)



The "running" state also includes "starting" tasks. Similarly, the "stopped" state also includes "stopping" tasks.

You can double-click the task counters to open the corresponding **Tasks** View.

Note: The task counters do not reflect tasks on servers that are not currently monitored. For more information, see [Additional server management options \(page 64\)](#).

- **For servers:** The total number of servers, the number of servers that are running and successfully monitored (green icon), the number of servers that Enterprise Manager cannot monitor due to connectivity issues (red icon), and the number of servers that are not being monitored (gray icon)

Status Bar Example



Monitoring Data Lake tasks

You can view general information on the progress of Compose Full Load or Change Processing operations in the **Task Dashboard**, located to the right of the task list.



*Use the arrow to collapse or expand the **Task Dashboard**.*

The top of the dashboard provides buttons for stopping and starting task(s), and viewing task log files. The following information about the task is also provided:

- The task name and state
- A link to the Compose project
- **Source:** The Landing Zone name and type in a Data Storage task or the project name and Landing Zone type in a Provisioning task.
- **Target:** The task name and target type in a Provisioning task or the project name and Landing Zone type in a Data Storage task.

The **Loading** tab in the right pane shows the following general progress information:

- **Load Duration:** The duration of the loading operation.
- **Ended:** When the loading operation completed.
- **Overall Progress bar:** The overall progress of the loading operation.
- **Table bars:** The status of the tables being loaded, which can be:

- **Completed** - The number of tables that have been loaded into the target.
- **Loading** - The number of tables that are currently being loaded into the target.
- **Queued** - The number of tables that are waiting to be loaded into the target.
- **Error** - The number of tables that could not be loaded due to an error
See [Messages and notifications \(page 269\)](#) and [Data error handling \(page 263\)](#) for more information about error messages.



*With Compose tasks, canceled tables are reflected in the **Total Tables** counter but not in the **Completed/Loading/Queued/Error** counters. Therefore, when one or more tables is canceled in a task, the sum of these counters will be less than the **Total Tables**.*

- The number of **Commands Completed** out of the **Total Commands**.

Monitoring Data Warehouse tasks

You can view general information on the progress of Compose Full Load or Change Processing operations in the **Task Dashboard**, located to the right of the task list.



*Use the arrow to collapse or expand the **Task Dashboard**.*

The top of the dashboard provides buttons for stopping and starting task(s), and viewing task log files. The following information about the task is also provided:

- The task name and state
- A link to the Compose project
- **Source:** The display name and type of the source database
- **Target:** The display name and type of the target database. Note that as the target database does not have a display name in Compose, the project name is displayed instead.

The **Loading** tab in the right pane shows the following task information:

- **Load Duration** - Load duration time
- **Ended** - Load end time
- Overall progress bar
- **<n> out of <n> commands completed** - The total number of ETL statements executed
- **Total Tables** - Total number of tables loaded
- **Total Records** - Total number of records loaded (also represented in a pie graph)
 - Total number of INSERT operations
 - Total number of UPDATE operations
- **Error Marts** (Not displayed for Data Mart tasks as Data Quality rules are run in the Data Warehouse only)
 - Total number of tables with errors
 - Total number of data errors

Data Warehouse task dashboard example

▶ Run ■ Stop 📄 View Logs >

📄 landing (Project: sqlserver)

Source: landing, Microsoft SQL Server Target: sqlserver, Microsoft SQL Server

Loading

Load Duration: 00:00:10 Ended: Jul 7, 2021 5:40:57 PM

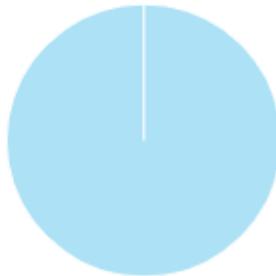


30 out of 30 commands completed

Total Tables 1

Total Records 61

Inserts ⁱ 61 (100%)
Updates 0 (0%)



Error Marts

Tables with errors 0
Data Errors

Monitoring workflows

In order to see your Compose workflows, expand the left pane, and make sure that the **Workflow Task Profile** is selected:

10 Monitoring and controlling tasks

Task Profile		Task List			
<input type="checkbox"/>	All	<input type="checkbox"/>	sm_comp...	big	Workflow_1
<input checked="" type="checkbox"/>	Data Mart (11)	<input type="checkbox"/>	sm_comp...	big	Workflow_2...
<input checked="" type="checkbox"/>	Data Storage (21)	<input type="checkbox"/>	sm_comp...	big	Workflow_f...
<input checked="" type="checkbox"/>	Data Warehouse (19)	<input checked="" type="checkbox"/>	sm_comp...	cdp2	Workflow_1
<input checked="" type="checkbox"/>	Provisioning (5)	<input type="checkbox"/>	sm_comp...	databricks	Default Cha...
<input checked="" type="checkbox"/>	Unidirectional (16)	<input type="checkbox"/>	sm_comp...	sql	Workflow_1
<input checked="" type="checkbox"/>	Workflow (20)	<input type="checkbox"/>	sm_comp...	sql	Workflow_2
		<input type="checkbox"/>	sm_comp...	sql	Workflow_3

To see which Compose project a workflow is associated with, expand the right pane to see the project name.

If you want to select other Compose task profiles (not just workflows), then it's a good idea to add the **Profile** column to the monitoring grid. This will allow you to easily differentiate between the different Compose task profiles, as shown in the following image:

State	Server	Project	Task	Profile
<input type="checkbox"/>	sm_comp...	test3	landing_CDC	Data Warehouse
<input type="checkbox"/>	sm_comp...	SqlServ...	northwind_CDC	Data Warehouse
<input type="checkbox"/>	sm_comp...	compos...	source_CDC	Data Warehouse
<input checked="" type="checkbox"/>	rb	Synapse	Default CDC ...	Workflow
<input type="checkbox"/>	rb	SQLProj...	Default Workfl...	Workflow
<input type="checkbox"/>	rb	Synapse	Default Workfl...	Workflow
<input checked="" type="checkbox"/>	rb	SQL1	Workflow_1	Workflow
<input type="checkbox"/>	rb	Synapse	Workflow_1	Workflow
<input checked="" type="checkbox"/>	rb	SQL1	Workflow_2	Workflow
<input type="checkbox"/>	shelly DL ...	spark	landing1	Data Storage
<input type="checkbox"/>	shelly DL ...	spark	landing2	Data Storage

For an explanation of how to add or remove columns, see [Customizing task columns \(page 249\)](#).

Limitations and considerations

- The **Stage** column will always show **Loading** regardless of the actual workflow stage. If you are not monitoring Replicate tasks (in which case this column might be useful), then it might be a good idea to hide it as described in [Customizing task columns \(page 249\)](#).
- The **Type** column will always show **Full Load & CDC** even if the workflow contains **Full Load** tasks only. This column is not displayed by default.
- While failed workflows are indicated with a  icon, workflows that completed successfully are indicated with a  icon. This might cause some confusion as the same icon () can also indicate that the workflow has not been run yet (which ceases to be an issue after it has been run once), or that it was stopped. Thus, the only way to be certain that a workflow completed successfully is by opening the Compose monitor as described in [Verifying that a workflow completed successfully \(page 247\)](#) below.

Controlling workflows and viewing logs



You can perform the action described below on several workflows simultaneously. To do this, you first need to select the workflows for which you want to perform the desired action and then either right-click the list and select the relevant option or click the relevant toolbar button.

Running and stopping workflows

You can run and stop workflows using any of the following methods:

- Right-click the workflow and select **Run** or **Stop** as appropriate.
- Select the workflow and click the **Run** or **Stop** toolbar buttons as appropriate.



*The **Run** button is not available for workflows that have not been validated or that contain elements that no longer exist in the Compose project.*

Viewing logs

You can view the workflow log files using any of the following methods:

- Right-click the workflow and select **View Logs**.
- Select the workflow and click the **View Logs** toolbar button.

See also: [Viewing and downloading log files \(page 286\)](#).

Verifying that a workflow completed successfully

As mentioned above, it is currently not possible to be certain that a workflow completed successfully just by looking at the **State**, as the  (stopped) state could also mean that the workflow was stopped manually or has not been run yet.

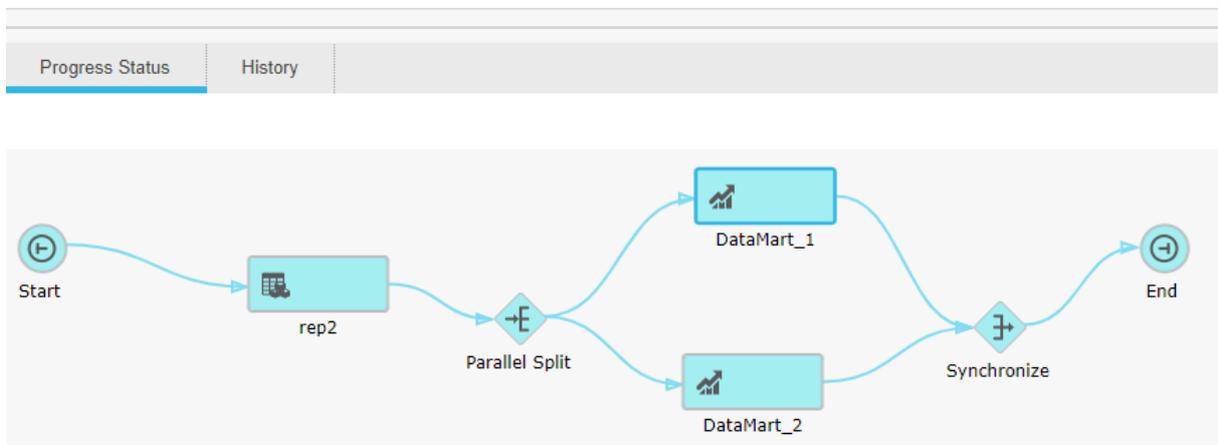
To verify that a workflow did indeed complete successfully, expand the right-pane and click the project name link.

10 Monitoring and controlling tasks

State	Server	Task	Stage	Tables with E...	Loading	Progress...	Target Thro...	Incoming
Stopped	rb	Workflow_New	Loading	0%	0	0	0	0

This will open the workflow in the Compose monitor, where you will be able to see its current state:

✓	Workflow_New	⇒ Workflow	4:28 PM	4:29 PM	N/A
---	--------------	------------	---------	---------	-----



10.3 Searching for tasks

You can search for specific tasks by typing a string into the **Search Tasks** box above the table. Enterprise Manager searches only textual columns, not numeric columns. The following columns are included in the search, even if a column is not displayed in the user interface:

- State
- Reason
- Server
- Task
- Profile
- Type
- Stage
- Loading Ended
- Source Name
- Source Type
- Target Name
- Target Type

- Tags
- Project (relevant for Compose tasks only)

You can also restrict a search to a specific column by preceding the search string with the column name and a colon, as follows: *ColumnName:value* (for example: *Server:john-vm.abc.local*). This is applicable to all available columns, not only the columns listed above.



When searching for a number, only enter whole numbers, no commas or special characters. For example, to search for 2,500, as displayed in the user interface, enter 2500; to search for 100%, enter 100.

10.4 Customizing task columns

You can choose to display or hide certain columns as well as sort columns as desired.

Column customization actions

To	Do this
Hide a column	<p>Right-click the heading of the column you want to hide and select Hide Column.</p> <p>Note that once you hide a column, if you want to display it again, you need to select it in the Column Settings dialog box (see below).</p>
Select which columns to display	<ol style="list-style-type: none"> 1. Right-click a column heading and select Column Settings. The Column Settings dialog box opens. 2. To display additional columns, in the left pane, select the columns you want to display and click the right arrow to move them to the right pane. 3. To hide columns, select them in the right pane and use the left arrow to move them to the left pane. 4. To change the column order, select a column in the right pane and use the up or down arrow to move it to the required position. 5. To reset column selection and order, click Restore Defaults. 6. Click OK.
Restore default columns	<ol style="list-style-type: none"> 1. Right-click a column heading and select Column Settings. The Column Settings dialog box opens. 2. To reset the column selection and order, click Restore Defaults. 3. Click OK.
Sort by column	<p>Click anywhere in the column heading to change the sorting order.</p> <p>OR</p> <p>Right-click the column and select Sort Ascending or Sort Descending as desired.</p> <p>An upward facing arrow indicates ascending order whereas a downward facing arrow indicates descending order.</p>

To	Do this
Export the list to a TSV file	Right-click any of the column headings and select Export to TSV file. When prompted, save the file in your desired location. Note that only currently displayed columns will be exported.

10.5 Grouping tasks

In the **Tasks** View, the left side (expanded by default) represents the **Tags** pane. It displays all available tags. Tags serve as attributes that you can attach to tasks to arrange them in Enterprise Manager. Tags allow you to group tasks that have a common denominator, and to filter by them, too. For example, you can organize tasks by flavor, such as all tasks that are part of a particular distribution process, by organization structure, by environment, by location, by application, by customer, and so on.

All tags:

- **Are static:** They remain constant during a task's life time. For example, the task type (CDC, FL, or both), source database type, or region of a task do not change as long as the task exists.
- **Are public:** All tags are available to all Enterprise Manager users.
- **Can be associated with any number of tasks:** You can associate a task with more than one tag.

Enterprise Manager comes with built-in tag categories as well as built-in tags. The following table lists all categories that are built in:

Task categories

Category	Tag
Custom	<ul style="list-style-type: none"> • Untagged
Task Type	<ul style="list-style-type: none"> • Full Load Only • CDC Only • Full Load & CDC
Task Profile	<ul style="list-style-type: none"> • Bidirectional • Unidirectional • Log Stream Staging - For more information about the Log Stream Staging feature, refer to the <i>Qlik Replicate Setup and User Guide</i>.
Source Database Type	If more than one source database type is available in your task list, one tag for each available source database type.
Target Database Type	If more than one target database type is available in your task list, one tag for each available target database type.
Replicate Servers	One tag for each managed Replicate server

10 Monitoring and controlling tasks

Category	Tag
Log Stream Staging	<p>One tag for each Log Stream Staging task. Selecting a tag in this category will show all tasks that are associated with the Log Stream Staging task (i.e. the Log Stream Staging task and all tasks that read from the same Log Stream staging folder).</p> <p>Selecting Other Tasks will show tasks that are not reading from Log Stream or tasks that are defined as Log Stream Staging tasks but that do not currently have a Log Stream target endpoint.</p>

Enterprise Manager automatically associates built-in tags with a task based on the task's characteristics, as derived from its definition. You cannot change the association of tasks with such tags.

In addition, you can create your own custom tags and assign tasks to them. You can delete such tags at any time. All tag names must:

- Be unique.
- Not exceed 64 characters.
- Only contain Latin characters, digits, spaces, dots (.), dashes (-), and underscores (_).
- Start with a digit or character.



Deleting a custom tag does not delete its associated tasks, and vice versa.

For each tag, Enterprise Manager displays the following information:

- The tag name
- The total number of tasks associated with this tag
- An icon that indicates when any associated task experiences a problem
If a task with a problem is associated with more than one tag, all corresponding tags indicate a problem.

The **Tags** column in the **Tasks** view presents a quick way to identify which tags are associated with each task.

The following table describes all options for viewing and managing tags.

Tag management options

To	Do this
Add a custom tag	In the Tags pane on the left, under Custom , in the New tag text box, enter a name for the tag and click the check mark to the right (or press Enter).

To	Do this
<p>Delete a custom tag</p> <p>Note: Deleting a tag does not delete the tasks associated with it.</p>	<p>In the Tags pane, under Custom, right-click the tag you want to delete and select Delete.</p> <p>When prompted for confirmation, click Yes.</p> <p>The tag is removed from the list.</p>
<p>Filter by tags</p>	<p>In the Tags pane, do one of the following:</p> <ul style="list-style-type: none"> • Select the check boxes of tags you want to include in the Tasks View. • Clear the check boxes of tags you want to exclude from the Tasks View. <p>When you select different tags:</p> <ul style="list-style-type: none"> • Within the same category, Enterprise Manager uses the logical OR operator. For example, if you select both Oracle and Netezza in the Target DB Type category, Enterprise Manager displays tasks to Oracle or Netezza targets. • In different categories, Enterprise Manager uses the logical AND operator. For example, if you select the Oracle tag in the Target DB Type category and the Finance tag in the Custom category, Enterprise Manager displays only task to an Oracle target that are also associated with the Finance tag.
<p>Assign custom tags to a task or remove tag assignments</p> <p>Note: This is only possible for custom tags. Built-in tags are assigned by default. Those assignments cannot be removed.</p>	<ol style="list-style-type: none"> 1. In the task list, select one ore more tasks to which you want to assign a tag and click Assign Tags. 2. In the Assign Tags window, select the required tag(s) or enter a new tag name in the New tag text box and click the check mark. 3. Click OK.

10.6 Running a task



Currently, this functionality is available for Replicate tasks only.

After you design a task (see [Defining and managing tasks \(page 69\)](#)), you can run and monitor its progress with one click in Enterprise Manager. This simple Click-2-Replicate function is described in this section. In addition, the various types of run options available are also described.

In this section:

- [How to run a task \(page 253\)](#)
- [Using the Run button options \(page 253\)](#)
- [Advanced run options \(page 254\)](#)
- [Recovering from data folder loss or corruption \(page 261\)](#)



The task run buttons area available in the toolbar at the top of the console in the following views:

- Tasks View (in both [Designer mode \(page 225\)](#) and [Monitor mode \(page 227\)](#))
- When [Viewing specific tasks \(page 225\)](#)

How to run a task

Click the **Run** button to execute a replication task. The task process continues to run until you click the **Stop** button to stop the task.



When you click **Run**, the following occurs:

- If this is the first time that a task is run, the **Start Processing** operation is run.
- If the task has been started and stopped, the **Resume Processing** operation described in [Using the Run button options \(page 253\)](#) is run.
- If changes were made to the endpoint, change processing takes place after the full load operation. If you do not want change processing to occur or if you want to start change processing from a predetermined point, you must make the appropriate [Advanced run options \(page 254\)](#) selection.

In some cases, task replication may stop due to an error although the task process is still running.

See [Task progress summary \(page 221\)](#) for information on the task status and how Qlik Replicate displays information on the current task status.

The **Run** button is available in the following views:

- The Tasks view when you select a task from the Task List.
- For the individual task, both the Designer mode and Monitor mode have the **Run** and **Stop** buttons available.



You must be in the Monitor mode to view the task progress.

Using the Run button options

Clicking the **Run** button runs a full-load replication task from the source to the target. This is a first time task that creates the target endpoints and loads the source data to the target according to your task definitions.

Subsequent runs allow you to resume processing from a specific point and process changes. In addition, you can also specify from what point you want the replication to start.

The following options are available:

- **Start Processing** - This is available the first time you run the task only. This will execute the initial full load operation. If **Change Processing** is also enabled for the task or if it is an Apply Changes only task type, change processing will start as soon as any changes are made to the source endpoint. (switches to **Resume Processing** after the task has started)
- **Resume Processing** - Resumes task execution from the point that it was stopped. You can also resume processing by clicking the **Run** button if the task has been stopped.



If the schema or a filter was changed after the task stopped, the task should be reloaded as opposed to resumed (see below).

- **Reload Target** (Only available when the **Full Load** or **Full Load** and **Apply Changes** replication options are enabled) - Starts the Full Load and Change Processing (if enabled) from the beginning. Tables that have already been processed are handled according to the [relevant "Target table preparation" setting](#).



To replicate tables that were added to the local file channel task after the initial full load, you need to reload both the local and the remote file channel tasks.

- **Advanced run options** - Advanced options for controlling task execution. For more information, see [Advanced run options \(page 254\)](#)

Start Processing

This is available the first time you run the task only. This will execute the initial full load operation. If **Change Processing** is also enabled for the task or if it is an Apply Changes only task type, change processing will start as soon as any changes are made to the source endpoint.

Reload target

Starts the Full Load and Change Processing (if enabled) from the beginning. Tables that have already been processed are handled according to the [relevant "Target table preparation" setting](#).



To replicate tables that were added to the local file channel task after the initial full load, you need to reload both the local and the remote file channel tasks.

Advanced run options

Advanced run options provide you with additional options for resuming and restarting tasks.



Some of the advanced run options are not available in a Log Stream Staging setup.

For information on the availability of advanced run options in a Log Stream Staging setup, refer to the Qlik Replicate Setup and User Guide.

To use Advanced Run Options, click the triangle next to the **Run** button and select **Advanced Run Options**.

The **Advanced Run Options** dialog box opens.

The **Advanced Run Options** dialog box lets you do the following:

- **Restart task and start processing changes from current time:** This starts the Apply Changes replication task from the beginning (as if the task has not run before).



Only available for Apply Changes replication tasks.

- **Tables are already loaded. Start processing changes from:**



Metadata changes performed on the source tables while a task is stopped (for example, DROP COLUMN) will not be captured when the task is resumed from an earlier time or position (SCN/LSN). In such a case, the metadata that exists when the task is resumed will be captured.

- **Date and Time:** Select the date and time from where you want to Replicate to start processing changes.



- *When logs are deleted from the database (e.g. due to a purge policy), a log matching the specified date and time may not exist. In this case, Replicate will resume the task from the earliest point possible, after the specified date and time.*
- *With the IBM DB2 for LUW source endpoint, this option cannot be used to start Apply Changes tasks from a point before the original start time of the Apply Changes task. Or, to put it another way, it can only be used to start tasks from any time after the original start time of the Apply Changes task.*
- *The timestamp uses the local time of the browser machine.*
- *This option is not relevant for the File Source endpoint.*

- **Source change position (e.g. SCN or LSN):** Specify the position in the log from where to resume change processing. The source change position format differs according to your source endpoint. For more information, see [How to Find the Source Change Position \(page 259\)](#).



The Source change position option is supported with the following source endpoints only:

- Oracle
- Microsoft SQL Server
- MySQL
- PostgreSQL
- IBM DB2 for z/OS
- IBM Informix
- IBM DB2 for LUW

Metadata Only:

The "Metadata only" options described below allow you to:

- Create empty tables on the target and then manually edit them.
- Create tables during a task.

Enabling the options will also ensure that supplemental logging is set up correctly on the source tables before starting the actual replication task.



The "Metadata only" feature is not supported when the task is defined with the Apply Changes only [task option](#). any of the following [task options](#):

- Apply Changes only
- Store Changes only
- Apply Changes and Store Changes

- **Recreate all tables and stop:** Select this option to recreate the target tables as defined in the **Full Load Settings** tab. When "Store Changes" is enabled, the Change tables/Audit table will be created as defined in the **Store Changes Settings** tab. To use this option, stop the existing task, run the task with this option enabled (the task will stop automatically) and finally, resume the task.
- **Create missing tables and stop:** Select this option to create missing target tables including Change Tables. You can use this option to create Change Tables on the target after enabling the "Store Changes" option (in the **Store Changes Settings** tab) for an existing task. To use this option, stop the existing task, run the task with this option enabled (the task will stop automatically) and finally, resume the task.

The table below shows which tables are created in the target database when the **Metadata only** option is enabled for a unidirectional task. As the table shows, when certain task options are enabled, [Control tables](#) and the [Audit table](#) will not be created on the target.

10 Monitoring and controlling tasks

Enabled Task Options	Tables Created on the Target
Full Load	All tables except for Control tables
Full Load and Apply Changes	All tables except for Control tables
Full Load, Apply Changes and Store Changes - When changes are stored in Change tables	All tables except for Control tables
Full Load and Store Changes - When changes are stored in an Audit table	All tables except for Control tables and the Audit table
Full Load, Apply Changes and Store Changes - When changes are stored in an Audit table	All tables except for Control tables and the Audit table

Recovery:

- **Recover using locally stored checkpoint:** Use this option if recovery is not possible using the **Resume Processing** or **Start process changes from** options (due to corrupt swap files, for example). When this option is selected, Replicate uses the checkpoint data stored in <Data_Folder_Path>\data\tasks\<<task_name>\StateManager to recover the task.



When using this option, the following limitations apply:

- The following source endpoints are supported only:
 - Amazon RDS for MySQL
 - Amazon RDS for PostgreSQL
 - Amazon RDS for SQL
 - AWS Aurora Cloud for PostgreSQL
 - Google Cloud SQL for MySQL
 - Google Cloud SQL for PostgreSQL
 - IBM DB2 for z/OS
 - Microsoft Azure SQL Managed Instance
 - Microsoft SQL Server
 - MySQL
 - Oracle
 - PostgreSQL
 - SAP HANA
 - The following target endpoints are supported only:
 - Google Cloud SQL for PostgreSQL
 - Microsoft Azure Database for MySQL
 - Microsoft Azure Database for PostgreSQL
 - Microsoft SQL Server
 - MySQL
 - Oracle
 - PostgreSQL
 - Tasks can only be recovered during Change Processing (i.e. after Full Load Completes)
 - With the exception of the File Channel endpoint, all target endpoints are supported. The following limitations apply:
 - In **Transactional apply** Change Processing mode: All target endpoints that support transactions are supported.
 - In **Batch optimized apply** Change Processing mode: Oracle target endpoint only is supported. Also requires the **Preserve transactional integrity** option to be enabled.
 - For all other target endpoints or Change Processing modes, recovery is supported, but may cause duplicates on the target.
- **Recover using checkpoint stored on target:** Select to recover a task using the CHECKPOINT value from the attrep_txn_state table (created in the target database).



When using this option, the following limitations apply:

- The following source endpoints are supported only:
 - Amazon RDS for MySQL
 - Amazon RDS for PostgreSQL
 - Amazon RDS for SQL
 - AWS Aurora Cloud for PostgreSQL
 - Google Cloud SQL for MySQL
 - Google Cloud SQL for PostgreSQL
 - IBM DB2 for z/OS
 - Microsoft Azure SQL Managed Instance
 - Microsoft SQL Server
 - MySQL
 - Oracle
 - PostgreSQL
 - SAP HANA
- Only the following target endpoints are supported:
 - Oracle
 - Microsoft SQL Server
- Tasks can only be recovered during Change Processing (i.e. after Full Load Completes)
- The task Change Processing mode must be set to either:
 - [Batch optimized apply](#) with the **Preserve transactional integrity** option enabled. Note that this mode is only supported with the Oracle target endpoint.
 - OR
 - [Transactional apply](#)For information about setting the Change Processing mode, see [Changes Processing Tuning](#).

This option will only be available if the **Store task recovery data in target database** option was enabled in the Task Settings' [Change Processing Tuning \(page 197\)](#) tab before Change Processing completed.

Select this option (as opposed to the **Recover using locally stored checkpoint** option) if the files in the **Data** folder are corrupted or if the storage device containing the **Data** folder has failed.

For a detailed explanation of how to set up and implement recovery using the `attrep_txn_state` table, see [Recovering from data folder loss or corruption \(page 261\)](#).

How to Find the Source Change Position

The following section explains how to locate the source change position for each of the supported databases.

MySQL

OPTION 1:

1. Execute:
`SHOW BINARY LOGS;`
2. Choose a binlog file (e.g. log.000123).
3. Execute:
`SHOW BINLOG EVENTS IN 'binlog_filename';`
Example:
`SHOW BINLOG EVENTS IN 'log.000123';`
4. Locate the position of a BEGIN or DDL statement (e.g. 1777888).



MySQL does not support a "start from position" located in the middle of a transaction, which is why you must select either BEGIN or DDL.

5. Set the internal parameter:
`startFromPosition = binlog_filename:begin_or_ddl_position`
Example:
`startFromPosition = log.000123:1777888`

OPTION 2:

1. Execute:
`SHOW MASTER STATUS;`
2. Set the "Start from position" as:
`firstColumnValue:secondColumnValue`
Example:
`mysql.007472:775`

Microsoft SQL Server

Execute:

```
SELECT MAX([Current LSN]) FROM fn_dblog(null,null);
```

Example: 00002717:00000e08:0003

Oracle

Execute:

```
SELECT current_scn FROM V$DATABASE;
```

Example: 1471212002

PostgreSQL

Execute:

```
SELECT * FROM pg_current_wal_lsn();
```

IBM DB2 for z/OS

Specify the LSN, which is the RBA in a non data sharing environment, and the LRSN in a data sharing environment

IBM DB2 for LUW

1. Execute `db2pd` to find the correct log name.
2. Execute `db2f1sn` with the returned log name and use the current LRI value.
For more information, refer to "Resuming or Starting a Task from LRI in a pureScale Environment" in the Replicate Help.

Recovering from data folder loss or corruption

During normal operation, Qlik Replicate maintains the replication state in the following location:

```
<Data_Folder_Path>\data\tasks\<task_name>\StateManager
```

This enables tasks that cannot be resumed normally (due to corrupt swap files, for example) to be recovered using the **Recover using locally stored checkpoint** option described in [Advanced run options \(page 254\)](#).

However, if the files in the **data** folder become corrupted or if the storage device containing the **data** folder fails, tasks must be recovered using the means described below.

This option is supported when replicating between the following endpoints only:

Supported source and target endpoints

Supported source endpoints	Supported target endpoints
<ul style="list-style-type: none">• Amazon RDS for MySQL• Amazon RDS for PostgreSQL• Amazon RDS for SQL• AWS Aurora Cloud for PostgreSQL• Google Cloud SQL for MySQL• Google Cloud SQL for PostgreSQL• IBM DB2 for z/OS• Microsoft Azure SQL Managed Instance• Microsoft SQL Server• MySQL• Oracle• PostgreSQL• SAP HANA	<ul style="list-style-type: none">• Oracle• Microsoft SQL Server

Setting up and initiating task recovery

For recovery to be successful, the source database transaction logs must be available from the time the task failed.

To set up a task for recovery

1. Design a task. Make sure to enable the **Store task recovery data in target database** option in the Task Settings' [Change Processing Tuning \(page 197\)](#) tab. This option can be enabled at any time during Change Processing, although it must be enabled before Change Processing completes.
2. Export the task definitions as described in the Qlik Replicate User Guide and Reference.
3. Run the task.

In addition to the selected source tables, the task will write the checkpoint data to the following table in the target database (and automatically create the table if it has not already been created by another task):

attrep_txn_state

To initiate recovery

1. Import the task definition exported when you set up the task.
2. Enter the passwords in the endpoint connection settings.
3. Access the `attrep_txn_state` table on the target database and locate the failed task in the `TASK_NAME` column. If there are tasks with the same name running on multiple Replicate Servers, you will also need to locate the appropriate server in the `SERVER_NAME` column. After locating the relevant task, copy the value in the corresponding `CHECKPOINT` column.
4. Select the **Recover using checkpoint stored on target** option and then provide the `CHECKPOINT` value (preferably by pasting) as described in [Advanced run options \(page 254\)](#).
5. Click **OK** to start the recovery.

During recovery, Replicate does not write anything to the target database until it identifies the commit event corresponding to the `CHECKPOINT` value. Once it identifies the `CHECKPOINT` commit event, recovery is performed and the task reverts to standard operation.

10.7 Error handling

The following sections describe how to best handle task and data errors:

- [Task error handling \(page 262\)](#)
- [Data error handling \(page 263\)](#)



The descriptions in this section are relevant to Replicate tasks only.

Task error handling

When a task fails with an error, **Tasks** View displays a red error icon in the State column. The following flow suggests a proper action path for handling task errors.

1. Move the pointer over the error icon to display a tooltip with a high-level error message and a time stamp.
2. For more information on the problem, do one of the following:

- Open the [Messages and notifications \(page 269\)](#) to view all messages for the task.
If you do not see any relevant messages, you may need to:
Select **Selected Item** to only view messages for the selected task.
Change the time frame.
The Message Center only displays the last 20,000 messages.
 - Click **View Logs** to find the error message in the log files.
 - Click **View Task** to see more detailed information about the current state of the task.
3. If possible, fix the error based on the information provided in the message.

For more information on error handling in Qlik Replicate, see the Qlik Replicate User Guide and Reference.

Data error handling

Data errors are errors related to data processing at the record level. When Qlik Replicate encounters a data error during replication, it issues an error warning. Enterprise Manager indicates when a task has data errors in any of its tables. For each task in the task list, Enterprise Manager shows its data error count, which is the sum of all data errors that occurred in the task's tables.

You can drill down to the task to view this information per table. You have the option to reset the counter per table when the data errors are no longer of interest to you. To view this information per task in the task list, you can display the **Data Errors** column. For information on column selection, see [Customizing task columns \(page 249\)](#).



Take the following into account:

- *Reloading a table automatically resets the corresponding counter.*
- *Resetting has no effect on the task's exception table.*
- *The error handling behavior of escalation handling for repeating data errors is affected by counter reset.*

10.8 Using the monitor tools

The monitor tools let you view additional information about the task. The following topics describe the information available through these tools:

- [Setting the logging levels \(page 264\)](#)
- [Viewing and downloading logs \(page 265\)](#)
- [Deleting logs \(page 265\)](#)
- [Downloading a diagnostics package \(page 266\)](#)
- [Downloading a memory report \(page 265\)](#)

Logging

In the **Logging <Task_Name>** window, you can set the logging level for the task you are currently monitoring as well as view, download, and delete log files.



- Any changes to the task logging levels take effect immediately without needing to restart the task.
- The logging level can also be set in the **Logging Level** sub-tab in the **Task Settings** dialog box. For more information, see [Logging \(page 210\)](#).

Setting the logging levels

To set logging levels:

1. Drill-down to Monitor View as described in [Viewing specific tasks \(page 225\)](#).
2. Then, click the **Tools** toolbar button and select **Logging**.
The **<Task_Name> Logging** window opens.
3. At the top of the **<Task_Name> Logging** window, set the **Component Logging Level** slider to the log level you want. This sets the logging level for all log modules. Note that all of the sliders for the individual modules move to the same position that you set in the main slider.
4. Make any changes to the sliders for the individual modules. This is optional. Note that if you change the main slider, all of the individual sliders are reset to the new position. If you want to maintain a different logging level for a specific module, you need to reset it.

Storing trace and verbose logging in memory



This option is relevant for Replicate tasks only and is not supported with the following source endpoints:

- MongoDB
- Salesforce (Streaming CDC)
- Salesforce (Incremental Load)
- SAP Application
- SAP ODP

When the logging level is set to "Trace" or "Verbose", you can instruct Replicate to store the logging information in memory until an error occurs. On detecting an error, Replicate will begin writing to the physical logs and continue to do so for a few minutes after the initial occurrence of the error.

If *no* error occurs before the allocated memory is used up, Replicate will empty the memory buffer and start afresh.

This option is useful for tasks that fail unpredictably and for no obvious reason. The problem with continually writing large amounts of information to the logs is twofold:

- Running in "Trace" or "Verbose" logging mode will quickly use up available disk space (unless the logging settings have been configured to prevent this).
- Continually writing large amounts of data to the logs will affect performance.

To use this option

1. Select the **Store trace/verbose logging in memory, but if an error occurs, write to the logs** check box at the top of the tab.
2. In the **Allocate memory up to (MB)** field, specify the amount of memory you want to allocate for storing logging information.

Viewing and downloading logs

To view and download logs

1. Drill-down to Monitor View as described in [Viewing specific tasks \(page 225\)](#).
2. Then, click the **Tools** toolbar button and select **Logging**.
The **<Task_Name> Logging** window opens.
3. Select the log file you want to view or download from the list in the **Log Files** pane. If you want to download the file, skip to [Step 7](#).
4. The contents of the log file will be displayed in the right pane. When you select a row in the log file, a tooltip will be display the full message of the selected row.
5. You can browse through the log file using the scroll bar on the right and the navigation buttons at the top of the window.
6. To search for a specific string in the log file, enter the search string in the search box at the top of the window.
Any terms that match the specified string will be highlighted blue.
7. To download the log file, click the  toolbar button.
Depending on your browser settings, one of the following will occur:
 - The task JSON file will be automatically downloaded to the default download location
 - You will be prompted for a download location. In this case, save the JSON file to your preferred location.

Deleting logs

To delete logs

1. Drill-down to Monitor View as described in [Viewing specific tasks \(page 225\)](#).
2. Then, click the **Tools** toolbar button and select **Logging**.
The **<Task_Name> Logging** window opens.
3. Click the **Delete Logs** button.
The **Delete Logs** window opens.
4. Optionally change the default number of days (45) and then click the **Delete** button.
All log files older than the specified number of days will be deleted.

Downloading a memory report

The memory report is a diagnostics tool that can be used to diagnose memory-related issues, such as unusually high memory consumption by a specific task.

Usually, multiple memory reports showing the gradual increase in memory consumption will need to be generated.

To download a memory report:

1. Drill-down to Monitor View as described in [Viewing specific tasks \(page 225\)](#).
2. Click the **Tools** toolbar button and then select **Support > Download Memory Report**. Depending on your browser settings, the following file will either be automatically downloaded to your designated download folder or you will be prompted to download it:
File name:
<task_name>__diagnostics__<timestamp>.memp
Example:
MyTask__diagnostics__20180109161333.memp
3. Send the report to Qlik.

Downloading a diagnostics package

You can generate a task-specific diagnostics package for Support to review. The diagnostics package contains the task log files and various debugging data that may assist in troubleshooting task-related issues.

To download a diagnostics package:

1. Drill-down to Monitor View as described in [Viewing specific tasks \(page 225\)](#).
2. Click the **Tools** toolbar button and then select **Support > Download Diagnostics Package**. Depending on your browser settings, the following file will either be automatically downloaded to your designated download folder or you will be prompted to download it:
File name:
<task_name>__diagnostics__<timestamp>.zip
Example:
MyTask__diagnostics__20180109161333.zip

10.9 Scheduling jobs

Use the Scheduler to schedule a one-time job or a recurrent job for specific Replicate task operations. A job is essentially an operation that can be scheduled to occur once, daily, weekly or monthly.

The following operations can be scheduled:

- Run/Resume a task
- Stop a task
- Reload a task



- *Changes to settings will only take affect after restarting all tasks.*
- *Compose tasks are not supported.*

To schedule a new job:

1. Switch to **Servers** view.

2. Click the **Scheduler** toolbar button.

The **Scheduler for <server_name>** window opens.

The window is divided into two tabs: **Scheduled Jobs** and **Executed Jobs**. The **Scheduled Jobs** tab contains a list of jobs that are scheduled to run periodically or once only while the **Executed Jobs** tab contains a list of jobs that have already run.



*The **Executed Jobs** tab will only show executed jobs that were scheduled to run once only. In other words, jobs scheduled to run periodically (e.g. Daily, Weekly, Monthly) will not be shown.*

3. Click the **New Scheduled Job** toolbar button.

The **New Scheduled Job** window opens.

4. Specify a **Job Name** and then, from the **Select scheduled job type** drop-down list, select one of the following:

- **Run task** to run or resume the task(s) at the scheduled time.



*For Full Load only tasks, it is preferable to select **Reload target** rather than **Run task** when the scheduling is set to Daily, Weekly or Monthly. This will update the table's data whereas **Run task** will replace the existing table.*

- **Stop task**
- **Reload target**



*Selecting **Reload target** will execute the task according to the task's replication settings. For example, if the task's **Full Load** and **Apply Changes** options are enabled, **Reload target** will reload the target tables and apply any subsequent changes.*

5. Select one of the following time conventions:

- **Use server local time** - When this option is selected (the default), the job will run when the specified time is reached in the server's location.
See also: [Impact of DST Change on Qlik Replicate](#).
- **Use universal time (UTC)** - When this option is selected, the job will run at the specified UTC time. So, for example, if the server is located in a UTC + 2 timezone, a job scheduled to run at 13:00 UTC time will actually run at 15:00 local server time. Scheduling a job to run in UTC mode may be useful if you need tasks on several Replicate servers (located in different timezones) to run concurrently.



*For reference, both the server local time and the UTC time are displayed to the right of the **Scheduled Time** heading.*

6. Select and define one of following scheduling options:

- **Once** - Run the job once on the specified day and at the specified time.
- **Daily** - Run the job every day at the specified time.
- **Weekly** - Run the job on the specified days and at the specified time.
- **Monthly** - Choose one of the following:
 - **Day** to run the job on the <nth> day of every month, and at the specified time.



*To run the job on the last day of every month, select **Last day of every month** from the drop-down list.*

- **The** to run the job on the <nth> <weekday> of every month, and at the specified time.
- **Every** - Run the job according to specified interval, starting on the specified date and time. Job intervals are always calculated according to the original start time. So, for example, if a job is scheduled to run every 3 minutes starting at 14:00 and the job completes at 14:05, then the next instance will run at 14:06.



The minimum interval is 5 minutes.

7. For the **Apply to tasks** option, select which tasks to schedule. Select either **All tasks** to apply the job to all current and future tasks or **Selected tasks** to apply the job to specific tasks. If you choose **Selected tasks**, a list of currently defined tasks is displayed. Select which tasks to apply the job to.
8. Click **OK** to save your settings.

To enable or disable a scheduled job:

- In the **Scheduled Jobs** tab, select or clear the check box in the **Enabled** column as required.

To edit a scheduled job:

1. Select the job in the **Scheduled Jobs** or **Executed Jobs** list.
2. Click the **Open** toolbar button and edit the job as required.

To delete a scheduled job:

1. Select the job in the **Scheduled Jobs** or **Executed Jobs** list.
2. Click the **Delete** toolbar button.

11 Messages and notifications

The Message Center is located at the bottom of the console and contains messages about monitored servers and tasks. This topic describes the various options for viewing and handling messages.

In this section:

- [Message Center overview \(page 269\)](#)
- [Customizing the Message Center \(page 271\)](#)
- [Viewing additional information \(page 272\)](#)
- [Notifications \(page 272\)](#)
- [Viewing and downloading log files \(page 286\)](#)

11.1 Message Center overview

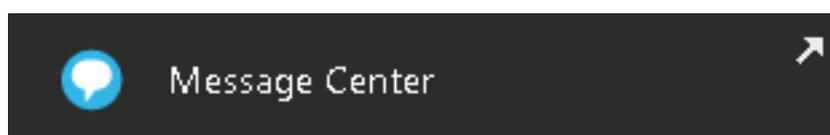
By default, the Message Center is minimized to a message tray at the bottom left of the console. The message tray notifies you of how many messages were received, during which time-period they were received, and the highest severity of all the messages (indicated by a color-coded callout icon). It is always global in scope, showing information for all servers and tasks.



The Message Center will not show log messages reported by Compose servers and tasks. However, messages reported by Enterprise Manager such as when monitoring has started /stopped for a Compose server or when a server has been deleted, will appear.

In the message tray, the message severity icon indicates the highest severity of the received messages. For example, if 11 messages were received but only one of them was an error message, the severity icon is red. Other callout colors are orange for warning messages and blue for informational messages.

Message Tray Example



To open or close the Message Center:

- Click the diagonal arrows to the right of the message summary.

To maximize the Message Center:

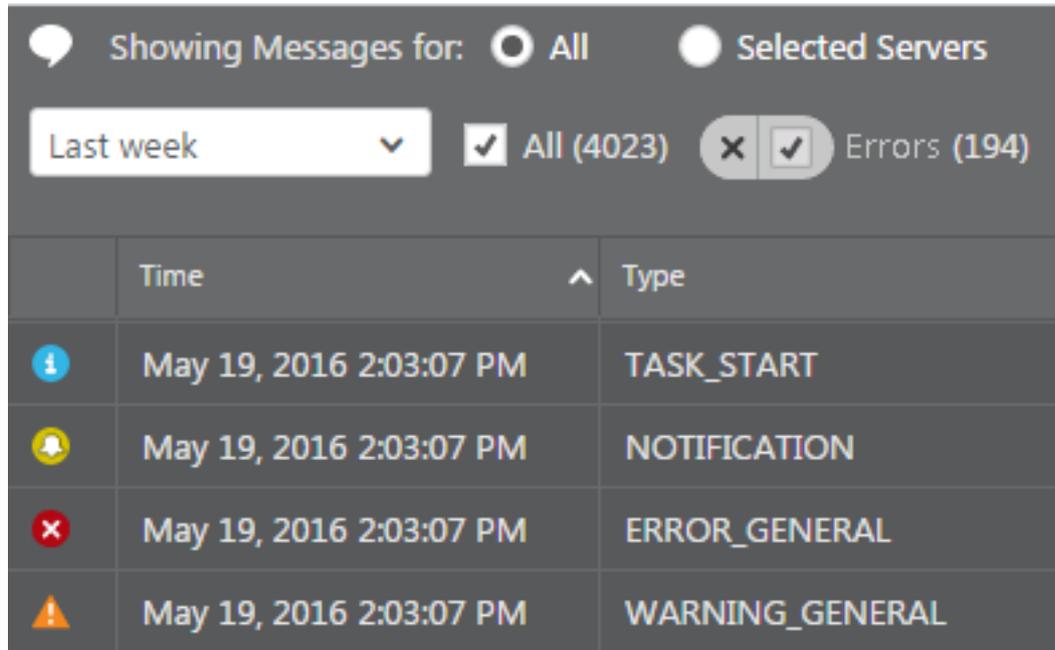
- When the Message Center is half-way open, click the Maximize icon in the top right corner.

To reduce the size of the Message Center:

- When the Message Center is fully open, click the Restore Down icon to the right of the message summary.

The following types of messages can be displayed: Info, Notification, Warning and Error. The actual message types that Qlik Enterprise Manager displays as well as the display time period depend on your Message Center preferences.

Each message type has its own icon, as shown below:



For each message, the following information is available:

- **Severity Icon:** Info, notification, warning or error
 - **Info:** circular blue "i" icon
 - **Notification:** circular yellow bell icon
 - **Warning:** triangular orange exclamation mark icon
 - **Info:** circular red "x" icon
- **Time:** When the event occurred
- **Reported By:** The display name of the Replicate server. For messages reported by Enterprise Manager, this field has a value of **Qlik Enterprise Manager**.
- **Server:** The name of the server in Enterprise Manager
- **Task:** The task that generated the message
- **Type:** The event that generated the message, such as TASK_START
- **Message:** The actual message as issued by the Replicate Server or Enterprise Manager

- **Error Code:** Shows the error code of task errors. You can set a task notification that will be sent whenever specific error codes are returned. For more information, see [Setting a task notification \(page 273\)](#).
Note that only error codes for tasks running on Replicate versions 6.2 or later will be shown.

Not displayed by default:

- **Table:** The table name when the message is related to a particular table. Will appear after the **Task** column if added.
- **ID:** A unique ID that serves as a reference number. You can copy the ID to the clipboard for easy reference, for example to paste it into an email when you need to refer to a specific message or to search for the message later. Will appear after the **Error Code** column if added.

11.2 Customizing the Message Center

You can customize the Message Center according to your business needs. The table below describes the available options.

Message Center customization options

To	Do this
Display messages for all tasks	Under Showing Messages for , select All .
Display messages only for the selected tasks or servers	Under Showing Messages for , select Selected Tasks or Selected Servers , respectively.
Hide current messages	Click Hide Current Messages above the message list to only show new messages coming in. Note that this option is only available when you view all system tasks. To bring back all messages, click Show All Messages .
Change the time period	From the drop-down list at the top left of the Message Center, select one of the available time periods: Last 6 hours , Last 12 hours , Last day , Last 3 days , or Last week .
Filter message severity	Select or clear the Errors , Warnings , Notifications or Info check boxes above the message list as desired.

The Message Center also includes buttons to view log files, open the Task tab to view a task, or view server information. In addition, you can:

- Customize the columns to display. For more information, see [Customizing task columns \(page 249\)](#).
- Configure the message purge policy. For more information, see [Message Center purge settings \(page 291\)](#).

Searching for messages

You can search for specific messages by typing a string into the **Search messages** box above the table. Note that Enterprise Manager searches only textual columns, not numeric columns. The following columns are included in the search, even if a column is not displayed in the user interface:

- Severity
- Reported By
- Server
- Task
- Message
- Error Code
- ID

You can also restrict a search to a specific column by preceding the search string with the column name and a colon, as follows: `columnName:value` (for example: `type:TASK_STOP`). This is applicable to all available columns, not only the columns listed above.

11.3 Viewing additional information

From the Message Center, you can directly open the task or server for a particular message or view its log file.

Message Center icon actions

Click	To View
	Log information: See Viewing and downloading log files (page 286) .
	Task information: See Monitoring Full Load replication (page 228) and Monitoring Change Processing replication (page 233) .
	Server information: See Monitoring servers (page 52) .



These buttons are only available at a certain window size. If the window is too small, they do not display. In this case, these options are only available from the context menu that appears when you right-click a table row.

11.4 Notifications

You can configure Enterprise Manager to notify you on the occurrence of a task and/or server event.



Notifications defined in Enterprise Manager do not conflict with notifications defined in Replicate. Although Replicate notifications cannot be managed via Enterprise Manager, Replicate notification messages will be shown in the Enterprise Manager Message Center.

Instructions for setting up notifications are provided in the following sections:

- [Setting a task notification \(page 273\)](#)
- [Setting a server notification \(page 280\)](#)

Setting a task notification



Task notifications apply to Replicate tasks only.

Notifications can be set for the following events:

- Processing status changes
- Excessive task latency
- Excessive memory usage
- Excessive disk usage
- Processing errors
- Other errors

To set a task notification:

1. Click the  toolbar button on the right of the console.
The **Notification Rules** window opens.
2. Click the **Tasks** tab and then click **New**.
The **New Task Notification** wizard opens.
3. Provide a name for the notification.
4. Select a notification in one of the available categories and, where applicable, set the time/size threshold for sending the notification.

Processing Status:

- **Task was started manually or by the Scheduler:** To send the notification when the task is started manually or by the Scheduler.
- **Task was stopped manually or by the Scheduler:** To send the notification when the task is stopped manually or by the Scheduler.
- **Task was stopped after Full Load: Cached changes were not applied:** To send the notification when the task is stopped after Full Load completes but before cached changes (changes to the source tables that occurred during Full Load) are applied to the target.
- **Task was stopped after Full Load: Cached changes were applied:** To send the notification when the task is stopped after Full Load completes and cached changes (changes to the source tables that occurred during Full Load) have been applied to the target.
- **Full Load started:** To send the notification when the Full Load process starts.
- **Full Load completed:** To send the notification when the Full Load process completes.

Performance/Resources:

- **Latency is higher than [N] seconds**
Specify the desired threshold value and unit of measurement (seconds or minutes).
- **Memory usage exceeds [N] GB**
Specify the desired threshold value and unit of measurement (GB or MB).
- **Disk usage exceeds [N] GB**
Specify the desired threshold value and unit of measurement (GB or MB).

Errors and Warnings:

- **Any Error:** Select this to receive a notification when any error occurs in the system. Note that notification will not be sent for the following error types:
 - Recoverable errors. Recoverable errors are printed as INFO in the log (unless the task logging is set to TRACE, in which case they will be printed as ERROR).
 - Certain user-defined error handling policies. For example, if the **Duplicate key when applying INSERT** Apply Conflicts handling option is set to **Log record to the exceptions table**, no notification will be sent. However, if it set to **Stop task**, a notification *will* be sent.



Setting this option may result in a large volume of notifications, depending on the number of tasks running and their current status. It is therefore recommended to define this notification for selected tasks rather than for all tasks.

- **Any Warning:** Select this to receive a notification when a warning is issued in the system.
- **No changes were captured in the last <n> minutes:** As this notification requires Replicate May 2022 or later, it will not be available if the monitored Replicate Server is an incompatible version. Select this notification if you want to be alerted about delays in capturing changes from the source database. With most databases, receiving this notification will usually indicate a connectivity problem. However, with the following databases, it could simply mean that no changes occurred during the specified period.
 - Microsoft SQL Server
 - IBM DB2 for LUW
 - IBM DB2 for iSeries

When the task resumes capturing changes, a notification that change capture has resumed will be sent, and the notification timer will be reset.

- **Table processing was suspended due to errors:** Select this to receive a notification when an error causes a table to stop processing during a full-load operation or suspend CDC. In this case, the table process stops, but the task continues.
- **Task stopped due to a non-recoverable error:** Select this to receive a notification when an error that cannot be retried is returned and the task is stopped due to this error.



An additional notification will be sent when the Performance/Resource threshold returns to normal (where "normal" is 10% below the defined threshold).

- **Errors containing one of these codes**

When this option is selected, a notification will be sent whenever an error matching one of the specified error codes occurs. Error codes for tasks that ended with an error are shown in the Message Center. You can copy the desired codes from the [Message Center](#) and paste them into this field.



- *Multiple error codes can be specified. In this case, error codes should be separated by a comma.*
- *Error codes must contain numeric values only.*

5. Click **Next**.

The **Associate** dialog is displayed.

6. Choose whether to associate the notification with **Selected Tasks** or with **All Current and Future Tasks**.

Note that when you complete the **Task Notification** wizard, the **Associated with** column in the **Notification Rules** window will display either **Selected Tasks** or **All Current and Future Tasks** depending on which of these options you selected.

If you chose **Selected Tasks**, from the **Server** drop-down list, select **All Servers** or a specific server. Then, select the desired tasks using the arrow buttons in the middle of the screen (multiple selection is supported).



The **Server** drop-down list only displays servers:

- *That are monitored*
- *For which the user who defines the notification has View permission*

The **Tasks** list only displays tasks:

- *That are running on monitored servers (that are not in an error state)*
- *For which the user who defines the notification has View permission*

7. Click **Next**.

The **Recipients** dialog is displayed.

8. Notifications will always be displayed in the Message Center. If you also want the notification to be sent to the **Windows Event Log** and/or specified **Email Recipients**, select their respective check boxes.

See also [Replicate Event IDs in Windows Event Log](#).

If you selected **Email Recipients**, specify a list of email recipients (separated by semi-colons) in the **To**, **Cc** and/or **Bcc** fields.



Sending notifications to email recipients requires your organization's outgoing [mail server settings](#) to be defined.

9. Click **Next**.

The **Message** dialog displays a default email subject and email/Windows Event Log message for the selected notification. You can change the default subject and/or message and make use of any of the [variables listed to the right of the message](#).



The message shown in the Message Center is system generated and cannot be edited.

- Click **Finish** to add the notification to Enterprise Manager.
A summary of the notification settings will be displayed in the **Tasks** tab in the **Notification Rules** window.



New notifications are always created as enabled and are active immediately without any need to stop/resume tasks.

Task notification variables

In addition to allowing you to edit the default messages, the **Message** window also provides a list of variables that you can insert into the notification message. These are described in the table below.

To insert a variable:

- Select the desired variable.
- Place the cursor in the message where you want the variable to be inserted.
- Click the arrow to the left of the variable list.



*The table below describes all of the message variables. However, the variables available for selection in the **Message** window will vary according to the notification event you select.*

Task notification variables

Variable	Description
<code>\${CDC_APPLIED_CHANGES_COUNT}</code>	The number of changes applied to the target tables during Change Processing.
<code>\${CDC_APPLY_LATENCY}</code>	The overall Change Processing latency.
<code>\${CDC_APPLY_THROUGHPUT_KB_SEC}</code>	The Change Processing throughput speed in kilobytes per second.
<code>\${CDC_APPLY_THROUGHPUT_REC_SEC}</code>	The Change Processing throughput speed in records per second.
<code>\${CDC_CHANGES_FOR_APPLY_IN_MEMORY_UNTIL_TARGET_COMMIT}</code>	The number of change records in memory waiting to be committed to the target database.
<code>\${CDC_CHANGES_FOR_APPLY_ON_DISK_UNTIL_TARGET_COMMIT}</code>	The number of change records on disk waiting to be committed to the target database.
<code>\${CDC_CHANGES_IN_MEMORY_UNTIL_SOURCE_COMMIT}</code>	The number of change records in memory until the next source commit.

11 Messages and notifications

Variable	Description
<code>#{CDC_CHANGES_ON_DISK_UNTIL_SOURCE_COMMIT}</code>	The number of change records on disk until the next source commit.
<code>#{CDC_COMMIT_CHANGE_RECORDS_COUNT}</code>	The total number of committed change records.
<code>#{CDC_COMMIT_CHANGE_VOLUME_MB}</code>	The total volume of committed change records in MB.
<code>#{CDC_COMMIT_TRANSACTIONS_COUNT}</code>	The total number of transactions committed during Change Processing.
<code>#{CDC_DDLS_COUNT}</code>	The number of DDLs (metadata) applied during Change Processing.
<code>#{CDC_DDLS_PERCENTAGE}</code>	The percentage of changes applied during Change Processing in terms of DDLs.
<code>#{CDC_DELETES_COUNT}</code>	The number of DELETES applied during Change Processing.
<code>#{CDC_DELETES_PERCENTAGE}</code>	The percentage of changes applied during Change Processing in terms of DELETES.
<code>#{CDC_INCOMING_CHANGES_COUNT}</code>	The number of incoming changes (from the source endpoint).
<code>#{CDC_INCOMING_TRANSACTIONS_COUNT}</code>	The number of incoming transactions (from the source endpoint).
<code>#{CDC_INSERTS_COUNT}</code>	The number of INSERTS applied during Change Processing.
<code>#{CDC_INSERTS_PERCENTAGE}</code>	The percentage of changes applied during Change Processing in terms of INSERTS.
<code>#{CDC_OPEN_TRANSACTIONS_IN_SOURCE}</code>	The number of open transactions in the source database during Change Processing.
<code>#{CDC_ROLLBACK_CHANGE_RECORDS_COUNT}</code>	The number of change records rolled back during Change Processing.
<code>#{CDC_ROLLBACK_CHANGE_VOLUME_MB}</code>	The volume of change records rolled back during Change Processing.
<code>#{CDC_ROLLBACK_TRANSACTIONS_COUNT}</code>	The number of transactions rolled back during Change Processing.
<code>#{CDC_SOURCE_LATENCY}</code>	The source latency during Change Processing.
<code>#{CDC_SOURCE_THROUGHPUT_KB_SEC}</code>	The Change Processing throughput from the source endpoint in kilobytes per second.
<code>#{CDC_SOURCE_THROUGHPUT_REC_SEC}</code>	The Change Processing throughput from the source endpoint in records per second.

11 Messages and notifications

Variable	Description
<code>\${CDC_TRANSACTIONS_FOR_APPLY_ON_TARGET}</code>	The number of transactions pending apply on the target
<code>\${CDC_UPDATES_COUNT}</code>	The number of UPDATES applied during Change Processing.
<code>\${CDC_UPDATES_PERCENTAGE}</code>	The percentage of changes applied during Change Processing in terms of UPDATES.
<code>\${TIME_SINCE_LAST_CHANGE}</code>	The time that has elapsed in minutes since changes were last captured from the source database.
<code>\${CPU_USAGE_PERCENTAGE}</code>	The percentage of CPU utilized by the task.
<div style="border: 1px solid black; padding: 5px;">  <p><i>Only available for Replicate 6.2 or later. For earlier Replicate versions, this variable will appear as -1 in the notification message.</i></p> </div>	
<code>\${DISK_USAGE_MB}</code>	The total disk usage in MB.
<code>\${EVENT_TIME}</code>	When the notification event occurred (e.g. when latency exceeded N MB, when latency returned to normal, and so on).
<code>\${FL_END_TIME}</code>	When the Full Load operation ended.
<code>\${FL_LOAD_DURATION}</code>	The duration of the Full Load operation.
<code>\${FL_PROGRESS_PERCENTAGE}</code>	The percentage of the Full Load operation already completed.
<code>\${FL_RECORDS_COMPLETED_COUNT}</code>	The current number of processed records during Full Load replication.
<code>\${FL_RECORDS_LEFT_COUNT}</code>	The current number of records awaiting processing during Full Load replication.
<code>\${FL_SOURCE_THROUGHPUT_KB_SEC}</code>	The Full Load throughput from the source endpoint in kilobytes per second.
<code>\${FL_SOURCE_THROUGHPUT_REC_SEC}</code>	The Full Load throughput from the source endpoint in records per second.
<code>\${FL_TABLES_COMPLETED_COUNT}</code>	The current number of tables loaded to the target during Full Load replication.
<code>\${FL_TABLES_LEFT_COUNT}</code>	The current number of tables still waiting to be loaded during Full Load replication.

11 Messages and notifications

Variable	Description
<code>\${FL_TABLES_LOADING_COUNT}</code>	The number of tables currently being loaded during Full Load replication.
<code>\${FL_TABLES_QUEUED_COUNT}</code>	The current number of queued tables during Full Load replication.
<code>\${FL_TARGET_THROUGHPUT_KB_SEC}</code>	The Full Load throughput to the target endpoint in kilobytes per second.
<code>\${FL_TARGET_THROUGHPUT_REC_SEC}</code>	The Full Load throughput to the target endpoint in records per second.
<code>\${FL_TOTAL_RECORDS_COUNT}</code>	The total number of records replicated during Full Load.
<code>\${FL_TOTAL_TABLES_COUNT}</code>	The total number of tables replicated during Full Load.
<code>\${FULL_LOAD_ENDED}</code>	A boolean indicating whether or not Full Load has completed.
<code>\${HOST_NAME}</code>	The host name of the Replicate Server machine.
<code>\${MEMORY_USAGE_MB}</code>	The total amount of memory being used.
<code>\${SERVER_NAME}</code>	The Replicate server display name, specified by the user when the server was added to Enterprise Manager.
<code>\${SOURCE_NAME}</code>	The name of the source endpoint.
<code>\${SOURCE_TYPE}</code>	The source endpoint type (e.g. Microsoft SQL Server).
<code>\${TABLES_WITH_ERROR_COUNT}</code>	The number of tables with errors.
<code>\${TAGS}</code>	The name of any tags applied in Enterprise Manager at the time of the notification.
<code>\${TARGET_NAME}</code>	The name of the target endpoint.
<code>\${TARGET_TYPE}</code>	The target endpoint type (e.g. Microsoft SQL Server).
<code>\${TASK_DATA_ERRORS_COUNT}</code>	The number of data errors encountered by the task.
<code>\${TASK_DESCRIPTION}</code>	The task description (entered by the user when the task was defined).
<code>\${TASK_NAME}</code>	The task name.
<code>\${TASK_PROFILE}</code>	The task profile (unidirectional or bidirectional).
<code>\${TASK_STAGE}</code>	The current processing stage of the task.
<code>\${TASK_STATE_REASON}</code>	The reason for the current task status.
<code>\${TASK_STATE}</code>	The current task status (stopped, running, etc.).

Variable	Description
\${TASK_TYPE}	The task type (Full Load only, Apply Changes only, or Full Load and Apply Changes)
\${NOTIFICATION_NAME}	The name of the notification.
\${ERROR_DETAILS}	Error details related to tasks that encountered an error. (currently relevant only for the "Task has stopped due to a non-recoverable error" event).

Setting a server notification

Notifications can be set for the following events:

- Excessive disk space usage (Replicate servers only)
- Excessive memory usage (Replicate servers only)
- State changes/errors (Replicate and Compose servers)



Currently, it is not possible to configure notifications for individual servers. In other words, any notification that you define will apply to all current and future servers.

To set a server notification:

1. Click the  toolbar button on the right of the console.
The **Notification Rules** window opens.
2. Click the **Servers** tab and then click **New**.
The **New Server Notification** wizard opens.
3. Provide a name for the notification.
4. Select and define one of the following:
In the **Disk Space** section, you can determine the disk usage event that triggers the notification.
Select one of the following:
 - **Disk usage reaches the high threshold:** The notification will be triggered when disk usage reaches [the percentage defined for the high threshold](#).
 - **Disk usage reaches the critical threshold:** The notification will be triggered when disk usage reaches [the percentage defined for the critical threshold](#).
 - **Disk usage returns to normal:** The notification will be triggered when disk usage returns to normal percentage (i.e. not high or critical).
 - **Disk usage reaches any of the defined thresholds or returns to normal:** The notification will be triggered in any of the following scenarios:
 - Disk usage increases from normal to the high threshold
 - Disk usage increases from normal to the critical threshold
 - Disk usage increases from the high threshold to the critical threshold
 - Disk usage returns to normal from the high threshold

- Disk usage returns to normal from the critical threshold
- Disk usage returns to the high threshold from the critical threshold

In the **System Memory** section, you can determine the system memory usage event that triggers the notification.

Select one of the following:

- **System memory usage reaches the high threshold:** The notification will be triggered when system memory usage reaches [the percentage defined for the high threshold](#).
- **System memory usage reaches the critical threshold:** The notification will be triggered when system memory usage reaches [the percentage defined for the critical threshold](#).
- **System memory usage returns to normal:** The notification will be triggered when system memory usage returns to normal percentage (i.e. not high or critical).
- **System memory usage reaches any of the defined thresholds or returns to normal:** The notification will be triggered in any of the following scenarios:
 - System memory usage increases from normal to the high threshold
 - System memory usage increases from normal to the critical threshold
 - System memory usage increases from the high threshold to the critical threshold
 - System memory usage returns to normal from the high threshold
 - System memory usage returns to normal from the critical threshold
 - System memory usage returns to the high threshold from the critical threshold

In the **State/Errors** section, select the **Server state has changed** notification to receive a notification whenever any of the following occurs:

- Enterprise Manager starts monitoring a Compose/Replicate server
- Enterprise Manager stops monitoring a Compose/Replicate server
- A connection error to any of the monitored Compose/Replicate servers occurs

5. Click **Next**.

The **Recipients** screen is displayed.

6. Notifications will always be displayed in the Message Center. If you also want the notification to be sent to the **Windows Event Log** and/or specified **Email Recipients**, select their respective check boxes. For more information about Windows Event Log IDs, see [Replicate Event IDs in Windows Event Log](#)

If you selected **Email Recipients**, specify a list of email recipients (separated by semi-colons) in the **To**, **Cc** and/or **Bcc** fields.



Sending notifications to email recipients requires your organization's outgoing [mail server settings](#) to be defined.

7. Click **Next**.

The **Message** screen displays a default email subject and email/Windows Event Log message for the selected notification. You can change the default subject and/or message and make use of any of the [variables listed to the right of the message](#).



The message shown in the Message Center is system generated and cannot be edited.

- Click **Finish** to add the notification to Enterprise Manager.
A summary of the notification settings will be displayed in the **Servers** tab in the **Notification Rules** window.



New notifications are always created as enabled and are active immediately without any need to stop/start monitoring servers.

Server notification variables

In addition to allowing you to edit the default messages, the **Message** window also provides a list of variables that you can insert into the notification message. These are described in the table below.

To insert a variable:

- Select the desired variable.
- Place the cursor in the message where you want the variable to be inserted.
- Click the arrow to the left of the variable list.



*The table below describes all of the message variables. However, the variables available for selection in the **Message** window varies according to the notification event you select.*

Variables related to Disk, Memory, Qlik CPU, and Machine CPU usage are not available for Compose servers. If defined, these variables will appear as -1 in the notification message.

Variables related to Qlik CPU and Machine CPU usage are only available for Replicate 6.2 or later. For earlier Replicate versions, these variables will appear as -1 in the notification message.

<code>\${ATTUNITY_CPU_USAGE_PERCENTAGE}</code>	Total percentage of CPU utilized by the Replicate Server's services and tasks.
<code>\${MACHINE_CPU_USAGE_PERCENTAGE}</code>	Total percentage of CPU utilized by the machine on which the server is installed.
<code>\${DISK_TOTAL_GB}</code>	The total size of the disk on which the Replicate "Data" folder is installed.
<code>\${DISK_USAGE_GB}</code>	The total amount of disk space(in GB) utilized by the Replicate "Data" folder.
<code>\${DISK_USAGE_MB}</code>	The total amount of disk space(in MB) utilized by the Replicate "Data" folder.
<code>\${DISK_USAGE_PERCENTAGE}</code>	The total percentage of disk space utilized by the Replicate "Data" folder.
<code>\${DISK_USAGE_PREV_STATE}</code>	The previous disk usage state (Normal, High, or Critical)

11 Messages and notifications

<code>\${DISK_USAGE_STATE}</code>	The current disk usage state (Normal, High, or Critical)
<code>\${ERROR_DETAILS}</code>	Information about server connection errors.
<code>\${ERROR_TASKS_COUNT}</code>	The total number of tasks in an "Error" state.
<code>\${EVENT_TIME}</code>	When the notification event occurred.
<code>\${HOST_NAME}</code>	The host name of the server machine.
<code>\${LAST_CONNECTION}</code>	The last successful connection to the server.
<code>\${LICENSE_DAYS_TO_EXPIRATION}</code>	The number of days left until the license expires.
<code>\${LICENSE_EXPIRATION}</code>	When the license is due to expire.
<code>\${LICENSE_ISSUE_DATE}</code>	When the license was issued.
<code>\${LICENSE_STATE}</code>	The current license state.
<code>\${MEMORY_TOTAL_GB}</code>	The total amount of memory available on the Replicate Server machine.
<code>\${MEMORY_USAGE_GB}</code>	The amount of memory (in GB) consumed by Replicate tasks on the Replicate Server machine.
<code>\${MEMORY_USAGE_MB}</code>	The amount of memory (in MB) consumed by Replicate tasks on the Replicate Server machine.
<code>\${MEMORY_USAGE_PERCENTAGE}</code>	The percentage of total memory consumed by Replicate tasks on the Replicate Server machine.
<code>\${MEMORY_USAGE_PREV_STATE}</code>	The previous memory state (Normal, High, or Critical)
<code>\${MEMORY_USAGE_STATE}</code>	The current memory state (Normal, High, or Critical)
<code>\${PLATFORM}</code>	The platform on which the server machine is installed (Linux or Windows)
<code>\${PORT}</code>	The server port.
<code>\${RECOVERING_TASKS_COUNT}</code>	The total number of tasks with a "Recovering" status.
<code>\${RUNNING_TASKS_COUNT}</code>	The total number of tasks with a "Running" status.
<code>\${SERVER_DESCRIPTION}</code>	The server description in Enterprise Manager.
<code>\${SERVER_NAME}</code>	The server name in Enterprise Manager.
<code>\${SERVER_PREV_STATE}</code>	The previous server state (Not Monitored, Monitored, or Error)
<code>\${SERVER_STATE}</code>	The current server state (Not Monitored, Monitored, or Error)
<code>\${STOPPED_TASKS_COUNT}</code>	The total number of tasks with a "Stopped" status.
<code>\${TASKS_TOTAL_COUNT}</code>	The total number of tasks, regardless of state.

<code>\${USER_NAME}</code>	The user name for accessing the server.
<code>\${VERSION}</code>	The server version.

Managing notifications

You can manage task and server notification in their respective tabs.

The following management options are available:



All of the actions described below can also be performed via the context menu.

Notification actions

To	Do this
Delete a notification	Select the desired notification(s) and then click the Delete toolbar button. When prompted to confirm your action, click Yes .
Edit a notification	Select the desired notification(s) and then click the Open toolbar button.
Enable a notification	Select the desired notification(s) and then click the Enable toolbar button.
Disable a notification	Select the desired notification(s) and then click the Disable toolbar button.
Search for a notification	Use the search box to perform a search on all the columns in the notification list. OR Search by column value by specifying the column name as a prefix. Example 1: To find all enabled notifications specify: Enabled: true Example 2: To find all notifications where "Jeff" is one of the recipients, specify: Recipients: Jeff

Required permissions

Notifications have their own set of permissions which are inherited from Enterprise Manager by default.

These are as follows:

- **Admin:** Can view notifications, enable/disable notifications, add/edit/delete notifications, and change the notification permissions.

- **Designer:** Can view notifications, enable/disable notifications, and add/edit/delete notifications.
- **Operator:** Can view notifications as well as enable/disable notifications.
- **Viewer:** Can only view notifications.

For more information on setting user permissions, see [Managing user permissions \(page 307\)](#)

Event IDs in Windows Event Log

The table below lists the Event IDs for tasks and server events in Windows Event Log.



Some events share the same ID. With these events, the recommended way of identifying the event is to parse the notification subject. This is especially recommended if you are using third-party tools to detect and report events.

Other Event IDs

Event ID	Description
261	Any error.
262	Any warning.
263	Errors containing the specified codes.

Server Event IDs

Event ID	Description
300	Server disk usage has changed.
300	Disk utilization has changed
300	Server disk usage has reached the high threshold.
300	Server disk usage has reached the critical threshold.
300	Server disk usage has returned to normal.
320	Server system memory usage has changed.
320	Memory utilization has changed.
320	Server system memory usage has reached the high threshold.
320	Server system memory usage has reached the critical threshold.
320	Server system memory usage has returned to normal.
340	Server state has changed.

Task Event IDs

Event ID	Description
----------	-------------

400	Task has started.
503	Full load has started.
504	Full load has completed.
403	Task has stopped after Full Load - cached changes were applied.
404	Task has stopped after Full Load - cached changes were not applied.
405	Task has stopped (other cases).
406	Task has stopped due to a non-recoverable error.
430	Task latency has exceeded the set limit.
431	Task latency is back to normal.
432	Task memory usage has exceeded the set limit.
433	Task memory usage is back to normal.
434	Task disk usage has exceeded the set limit.
435	Task disk usage is back to normal.

Table Event IDs

Event ID	Description
502	Table processing was suspended due to errors.

11.5 Viewing and downloading log files

In Enterprise Manager, you can view different types of log files:

- **Replicate Server log files (repsrv.log):** Retrieved from the monitored Replicate server.
- **Replicate Task log files (reptask_<task_name>.log):** Retrieved from the monitored Replicate server.
- **Compose Server log files (Compose.log):** Retrieved from the monitored Compose server.
- **Compose Agent log files (Compose.log):** Retrieved from the monitored Compose server. When selecting a task associated with a Compose for Spark project, you can also view the Compose job files in the Spark History Server as described below.
- **Compose task log files (<n>.log):** Retrieved from the monitored Compose server.
- **Operations log files (EnterpriseManager.log):** Created by and managed in Enterprise Manager.
- **Analytics log files (aemanalytics.log):** Created by the Enterprise Manager Analytics processes.

To view a log file:

1. Do one of the following to open the Log Viewer:

- For server log files: In the **Servers** list, select a server and click **View Logs** above the list.



This option is only available for servers that are connected.

- For Compose and Replicate task log files: In the **Tasks** list, select a task and click **View Logs** above the list.



This option is only available for tasks associated with connected servers.

When selecting a task associated with a Compose for Spark project, a **View Spark History Server** link appears at the bottom of the window. Clicking the link will open a window displaying the Compose Spark jobs.



*If you encounter an access error when clicking the **View Spark History Server** link, try one or both of the following:*

- Add an entry to the client host file that maps the Spark History Server host name to its externally accessible IP address.
- Open the necessary firewall ports to allow Enterprise Manager to access the Spark History Server.

- For Operations log files: Click the **View Logs** icon in the top right corner. Then select **Operations** from the **Log Files** drop-down list in the top left of the window.



*You can also view log files for messages in the Message Center. The **View Logs** option is available for all messages reported by Enterprise Manager, for messages reported by a server that is connected, and for tasks pertaining to a server that is connected.*

- For Analytics log files: Click the **View Logs** icon in the top right corner. Then select **Analytics** from the **Log Files** drop-down list in the top left of the window.
2. In the **Log Viewer** window, in the **Log Files** pane on the left, select the log file you want to view. The content of the log file is displayed in the right pane. When you select a row in the log file, a tooltip displays the full message of the selected row.
 3. Browse through the log file using the scroll bar on the right.
 4. To search for a specific string in the log file, enter the search string in the search box at the top of the window. Any terms that match the specified string are highlighted blue. The number of matches is displayed next to the search box. You can use the navigation errors to move to the first, next, previous, or last occurrence of the search string.
 5. To start a new log file, click **Roll Log File**.

The log file gets saved with a 12-digit timestamp appended to its name, such as *EnterpriseManager_160619073410.log* or *reprsv_160703131920.log*.



This option is not available for Compose log files.



*This option is not available when you open the **Log Viewer** window from the Message Center.*

6. Click **Close**.

To download a log file:

- In the **Log Viewer** window, select the log file you want to download and click the **Download Log File** icon at the top right.

12 Administration

This section describes how to configure message purge and logging settings in Qlik Enterprise Manager.



For information on setting up and using Analytics, see [Analytics \(page 325\)](#).

In this section:

- [Enterprise Manager settings \(page 289\)](#)
 - [Enterprise Manager logging settings \(page 289\)](#)
 - [Message Center purge settings \(page 291\)](#)
 - [Repository connection settings \(page 292\)](#)
 - [Qlik Catalog Server connection \(page 293\)](#)
 - [Analytics - Data collection and purge settings \(page 293\)](#)
 - [Configuring mail server settings \(page 296\)](#)
 - [Registering and managing licenses \(page 296\)](#)
- [User permissions \(page 298\)](#)
- [Creating an audit trail \(page 309\)](#)

12.1 Enterprise Manager settings

In the **Settings** window you can configure the following:

- [Enterprise Manager logging settings \(page 289\)](#)
- [Message Center purge settings \(page 291\)](#)
- [Repository connection settings \(page 292\)](#)
- [Analytics - Data collection and purge settings \(page 293\)](#)
- [Configuring mail server settings \(page 296\)](#)
- [Registering and managing licenses \(page 296\)](#)

To open the **Settings** window, click the  toolbar button in the top right of the console.

Enterprise Manager logging settings

In Enterprise Manager, you can modify the logging settings for system log files. This includes specifying:

- [Logging levels \(page 290\)](#)
- [Automatic log rollover and cleanup \(page 290\)](#)

To modify logging settings for server and task log files, you need to access the web console for the respective monitored Replicate server.

Logging levels

The logging level you set determines what information is written to the system log file, *EnterpriseManager.log*, and whether information is written to the *Attunity.WebLog.log* file. The system log file provides information about events, warnings, and errors occurring in Enterprise Manager, but not for the monitored Replicate servers and their respective tasks. The WebLog file captures requests and responses between the client and server, but only when the logging level for the WebLog component is set to Trace or Verbose.



*Because the WebLog file grows quickly in size, it is recommended that you only set the logging level for the WebLog component to **Trace** or **Verbose** for short periods of time, such as when troubleshooting an issue.*

The following logging levels are available, from the lowest to the highest:

- **Error:** Include only error messages.
- **Warning:** Include error and warning messages.
- **Info:** Include error, warning, and info messages.
- **Trace:** In addition to error, warning, and info messages, include debug data.
- **Verbose:** In addition to error, warning, and info message, include detailed debug data.

Within the log file, the logging level is indicated by the initial letter: E for error, W for warning, and so on. The higher levels always include the messages from the lower levels. Therefore, if you select Error, only error messages are written to the log. However, if you select Info, informational messages, warnings, and error messages are included. Selecting Verbose writes all possible messages to the log.

You can set a global logging level for all log components or separate logging levels for each component. For example, you can define a logging level of Info for Message Center logs and a logging level of Warning for Replicate tasks.

To set the logging level:

1. In the top right corner, click the gear icon.
2. In the **Settings** window, in the **Logging Levels** tab, move the top slider to the log level you want. This sets the log level for all components. Note that the sliders for all components move along to the same position.
3. Optionally, modify the individual logging level for any component.
4. Click **OK**.

Automatic log rollover and cleanup

You can define when Enterprise Manager should roll over the system log file and WebLog file and when to purge old log files. The current log files are called *EnterpriseManager.log* and *Attunity.WebLog.log*, respectively. Rolled over log files have a 12-digit timestamp appended to their name, such as *EnterpriseManager_160407111842.log* or *Attunity.WebLog_160717115348.log*.

Automatic rollover is enabled by default. You can also perform rollover manually if needed. See [Manual log rollover \(page 291\)](#).

To configure automatic rollover and cleanup:

1. In the top right corner, click the gear (settings) icon.
2. In the **Settings** window, in the **Logging|Log File Management** tab:
 - Select the **Enable automatic rollover** check box to have log files rolled over at the default size of 100 MB. By default, this check box is selected.
 - If you prefer a different cutoff size for rollover, select the check box **Roll over the log if the log file is larger than (MB):** and specify a different file size. When the log file reaches the specified size, the old log is saved with a timestamp appended to its name and a new log file is started. The maximum file size is 1,024 MB.
 - Under **Maximum number of newest log files to keep**, specify the maximum number of log files to keep. By default, this is 45 files. The maximum number of files cannot exceed 100. Enterprise Manager keeps the newest log files and removes any files beyond the specified number.
3. Click **OK**.

Manual log rollover

If you need to start a new system log file or WebLog file before the current file has reached the size specified for automatic rollover, or if you want to start a new server or task log file, you can do this manually. Manual rollover is only available for active log files and for log files pertaining to tasks that are currently running.

To manually roll over a log file:

1. In the top right corner, click the **View Logs** icon.
2. In the **Log Viewer** window, select the current log file and click **Roll Log File**. Enterprise Manager starts logging to a new log file. Rolled over log files have a 12-digit timestamp appended to their name, such as *EnterpriseManager_160407111842.log* or *Attunity.WebLog_160717115348.log*.
3. Click **Close**.

Message Center purge settings

Enterprise Manager pulls error, warning, information, and notification messages from all managed Replicate Servers and stores them locally on the machine where Enterprise Manager is installed. The Enterprise Manager console then retrieves these messages from the local storage and displays them in the Message Center as described in [Messages and notifications \(page 269\)](#).

To avoid storage issues, you can specify the interval at which Enterprise Manager should purge messages from the Message Center and the maximum number of messages to be stored. By default, Enterprise Manager purges messages after 14 days and when the number of messages exceeds 200,000. Purged messages are no longer available for retrieval.



The message purge policy you define in Enterprise Manager does not affect any log purge policy of the monitored Replicate servers.

To modify Message Center purge settings:

1. Click the gear icon in the top right corner.
The **Settings** window opens.
2. In the **Message Center** tab, specify:
 - After how many days or weeks to purge messages. The default value is 14 days; the maximum value 91 days (13 weeks). You can enter any value between 1 and 91.
 - The maximum number of messages to be stored in Enterprise Manager. The default value is 200,000. The maximum value is 1,000,000; the minimum 100. You can enter any value between 100 and 1,000,000.



*Enterprise Manager purges messages at 12:00 PM and 00:00 AM only. As such, it is possible that the number of actual messages may occasionally exceed the value specified in the **Store no more than** field.*

3. Click **OK**.

Repository connection settings

The repository is used to store data from the Analytics module. Once PostgreSQL is installed, you need to configure the connection settings to the repository.

To set up connectivity:

1. Switch to **Operations** View and then click the **Settings** button in the top right corner.
2. In the **Settings** window, click the **Repository Connection** tab.
3. Enter the following information:

Repository connection fields

Field	Value
Repository host	The IP address or host name of the PostgreSQL machine.
Port	The port through which the PostgreSQL machine is accessed.
Database	The name of the PostgreSQL database where you want the analytics data to be stored.
User name	The user name for accessing the PostgreSQL database.
Password	The password for accessing the PostgreSQL database.
Maximum number of connections	The number of concurrent connections to the specified PostgreSQL database. You may need to increase the number of connections based on task activity.

Field	Value
Additional connection properties	Enter any additional connection properties for the PostgreSQL database. Properties should be formatted as follows: "key=value;key=value" For a description of JDBC connection properties, visit: https://jdbc.postgresql.org/documentation/use/

- To test the settings (recommended), click **Test Connection**.
- If the connection test is successful, click **Save**. Note, after clicking **Save**, if you have specified a new database, you will also need to click **Initialize Analytics Repository** to enable the Analytics module.



*Clicking **Initialize Analytics Repository** will delete all data from an existing Repository. Therefore, you should only initialize them if you are configuring connectivity to a new database.*

- Configure the data collection and purging settings for Analytics as described in [Analytics - Data collection and purge settings \(page 293\)](#).

Qlik Catalog Server connection

For information on these settings, please refer to [Cataloging tasks in Qlik Catalog \(page 320\)](#).

Analytics - Data collection and purge settings

When working with the Analytics module, after [configuring the connection settings to your repository](#), you then need to configure data collection and purge settings.

You can also change the log level and management settings if desired.

Defining data collection and purging settings



Each time you add a Replicate server to be monitored, you need to stop and start the collector as described below. For information about adding servers, see [Adding Servers \(page 50\)](#)

To set up and start the collector:

- Select the **Analytics** tab.
- In the **Collector properties** section, optionally change the following information:

Collector property fields

Field	Value
Collect data every	The number of minutes to wait between data collection operations.
Store data up to	The number of days to store data from the moment it is collected.
If the repository is unavailable, store data in memory for up to	The number of minutes of data that should be stored in memory if the Analytics repository is unavailable for any reason.
Analyze database tables during data collection	When selected (the default), the Collector process will analyze the tables in the Analytics repository during each run. If unchecked, the Collector process will not analyze the tables and the customer should ensure that statistics are being updated periodically. As the size of the database gets larger, it is less important to update statistics frequently and it may start to impact the performance of the Collector, so it is recommended to disable if you notice the Analytics dashboards being updated less frequently.

3. Click **Start Collector**.

Stopping the collector

To stop the collector:

- Click the **Stop Collector** button in the **Collector properties** section.



*To resume data collection, click the **Start Collector** button.*

Stopping and starting the purger

To start and stop the purger:

- Click the **Start Purger** button. Data will be purged according to the value specified in the **Store data up to** field described above.
To prevent data from being purged, click the **Stop Purger** button.

Setting logging levels

In the **Logging Levels** sub-tab, you can set the logging levels for the log files generated by the Analytics service. The level of information in the log files is determined by the logging level set in this tab. Analytics log files can be viewed in the Log Viewer as described in [Viewing and downloading log files \(page 286\)](#).



Because the WebLog file grows quickly in size, it is recommended that you only set the logging level for the WebLog component to **Trace** or **Verbose** for short periods of time, such as when troubleshooting an issue.

The following logging levels are available, from the lowest to the highest:

- **Error:** Include only error messages.
- **Warning:** Include error and warning messages.
- **Info:** Include error, warning, and info messages.
- **Trace:** In addition to error, warning, and info messages, include debug data.
- **Verbose:** In addition to error, warning, and info message, include detailed debug data.

Within the log file, the logging level is indicated by the initial letter: E for error, W for warning, and so on. The higher levels always include the messages from the lower levels. Therefore, if you select Error, only error messages are written to the log. However, if you select Info, informational messages, warnings, and error messages are included. Selecting Verbose writes all possible messages to the log.

You can set a global logging level for all log components or separate logging levels for each component. For example, you can define a logging level of Info for Message Center logs and a logging level of Warning for Replicate tasks.

To set the logging level:

1. Move the top slider to the log level you want.
This sets the log level for all components. Note that the sliders for all components move along to the same position.
2. Optionally, modify the individual logging level for any component.
3. Click **OK**.

Automatic rollover and cleanup

You can define when Enterprise Manager should roll over the Analytics log file and when to purge old log files. The current log file is called *aemanalytics.log*. Rolled over log files have a 12-digit timestamp appended to their name, such as *aemanalytics_180407111842.log*.

Automatic rollover is enabled by default. If needed, you can also perform rollover manually as described in [Viewing and downloading log files \(page 286\)](#).

To configure automatic rollover and cleanup:

1. Select the **Log File Management** sub-tab.
2. In the **Settings** window, in the **Logging|Log File Management** tab:
 - Select the **Enable automatic rollover** check box to have log files rolled over at the default size of 100 MB. By default, this check box is selected.
 - If you prefer a different cutoff size for rollover, select the check box **Roll over the log if the log file is larger than (MB):** and specify a different file size. When the log file reaches the

specified size, the old log is saved with a timestamp appended to its name and a new log file is started. The maximum file size is 1024 MB.

- Under **Maximum number of newest log files to keep**, specify the maximum number of log files to keep. By default, this is 45 files. The maximum number of files cannot exceed 100. Enterprise Manager keeps the newest log files and removes any files beyond the specified number.

3. Click **OK**.

Configuring mail server settings

Mail server settings are required for sending Enterprise Manager notifications.

To configure the mail server settings:

1. Click  toolbar button in the right of the console.
The **Settings** window opens.
2. Select the **Mail Server Settings** tab and enter the following information:
 - **Mail server:** The host name or IP address of your organization's outgoing mail server.
Example: `smtp.example.com`
 - **Port:** The port used to communicate with the mail server. The default value is 25.
 - **Use SSL:** Select to connect to the mail server (for sending notifications) using SSL.
 - **Verify peer:** Select if you want the mail server to send a certificate proving its identity. On Windows, the certificate will be authenticated using the default CA certificates.
 - **CA Path:** On Linux, specify the location of the server CA certificate.
 - **Verify host:** Select to verify that the host name in the certificate matches the host name of the mail server.
 - **Anonymous login:** Select this to allow users to receive notifications without having to provide login credentials.
 - **User name:** The user name for the email user account that is sending notifications.
 - **Password:** The password for the email user account that is sending notifications.
 - **Sender email address:** The email address from which notifications will be sent. The address will appear in the **From** field of the email notification.
3. To send a test mail, specify an email address in the **Send to** field and then click the **Send Test Mail** button.
If the mail server settings are correct, an email will be sent to the specified email recipient. Otherwise, an error will be shown at the top of the console.
4. Click **OK** to save your settings.

Registering and managing licenses

In the **Licenses** tab, you can view and register licenses for each of the Qlik Enterprise Manager modules. Currently, the following modules require a license:

- **Replication Management** - Enables [design](#), [customization](#), [monitoring](#), and [control](#) of Replicate tasks as well as [Replicate Server management](#).

- **Replication Analytics** - Provides [measurements of server and task metrics](#) over a specific time-period.



When the Replication Analytics license expires, data collection will no longer occur. However, the Analytics dashboards will continue to show the analytics that were collected before the license expired. Additionally, the [Analytics tab](#) will also remain available.

Registering a license

To register a license:

1. Do one of the following:
 - Click the **Register License** link in the **Register License** message that is displayed when you open Qlik Enterprise Manager, or that is displayed in the main **Analytics** tab (after you have already registered a Replication Management license). The **Register <Module Name> License** window opens. Continue from [Step 5 below](#).
 - Click the  toolbar button in the right of the console.
2. The **Settings** window opens.
3. Select the **Licenses** tab.

The following sub-tabs are displayed (one for each module):

 - Replication Management
 - Replication Analytics
4. To register a license, select the relevant tab, and then click the **Register License** button at the top of the tab.

The **Register <Module Name> License** window opens.
5. Click **Load** to load the license from a file, or copy the license text (e.g. from an email message) into the **License text** area.



*If the license you loaded or pasted is for a different module (e.g. you intended to load a Replication Management license, but mistakenly loaded a Replication Analytics license), a warning will appear at the top of the **Register <Module Name> License** window.*

In such a case, you can either register the license anyway (since you may have intended to register that license later) or load/paste another the license (i.e. the license that you originally intended to register).

6. Click the **Register License** button at the bottom of the window.

If the license is valid, a confirmation that the license was registered successfully will be shown at the top of the **Settings** window. Otherwise, an appropriate error message will be shown.

Viewing licenses

You can view the properties of registered licenses (such as when the license is due to expire).

To view a license:

1. Click the  toolbar button in the right of the console.
The **Settings** window opens.
2. Select the **Licenses** tab.
3. Select the **Replication Management** or **Replication Analytics** sub-tab accordingly.
A list of properties will be shown.

Showing/hiding the Analytics Dashboards

The main **Analytics** tab is shown by default, regardless of whether or not you have registered a **Replication Analytics** license. If you do not intend to use the **Replication Analytics** module, you can hide the tab by either clicking the **Hide Analytics Dashboards** link in the **Analytics** tab or by following the procedure described below.

To show/hide the main Analytics tab:

1. Click the  toolbar button in the right of the console.
The **Settings** window opens.
2. Select the **Licenses** tab.
3. Select the **Replication Analytics** sub-tab.
4. Clear or select the **Hide Analytics Dashboards** check box as required.



This option is only available when no Replication Analytics license has been registered.

5. Click **Save**.

License alerts

If any of the Enterprise Manager licenses are invalid, due to expire, or have already expired, a **License Alerts** link will appear at the top of the console. Clicking the link will list the issue(s) and provide links to resolve them (e.g. Register License).

Permissions

For information on permissions required to perform the various the license-related operations (i.e. registering licenses, viewing licenses, and hiding the **Analytics** tab) see [Roles and permissions \(page 304\)](#).

12.2 User permissions

You can grant Qlik ReplicateEnterprise Manager users different roles according to the tasks you want them to perform. Qlik ReplicateEnterprise Manager comes with the following predefined security roles: Admin, Designer, Operator and Viewer. Each role has its own set of permissions, as described in [Roles and permissions \(page 304\)](#). For more information on permissions, see [Granular access control \(page 300\)](#)

You can associate a user with a security role by adding the user to the appropriate Active Directory group or by assigning a role directly to the user. By default, the user under whose account you install Enterprise Manager is associated with the Admin role. In addition, you can fine-tune access control per user or group. For more information, see [Granular access control \(page 300\)](#).

As a user with the relevant permissions, you can view and change the permissions for existing users or groups, or add users or groups that do not yet exist in Enterprise Manager.

The advantage of adding groups over users is that you can assign a security role to a group as a whole, instead of to individual users, and any new user that gets added to an existing group automatically acquires the security role granted to that group.

You can also:

- Add users as domain users that are not related to an active directory group.
- Add local groups. However, this is not possible by default. To set up Enterprise Manager to work with local groups, perform the steps in [Working with local groups \(page 306\)](#).

To set user permissions using Active Directory groups, you can either create Active Directory groups with the names listed in the table below, or you can create Active Directory groups with different names. Then, add users to the groups according to the role you want them to perform.

If you create your own Active Directory groups, you need to add them to the **User Permissions** tab in the **Settings** window and set their permissions as described in [Managing user permissions \(page 307\)](#).

Roles and AD groups

Role	Active Directory Group
Administrator	AttunityEnterpriseManager Admin
Designer	AttunityEnterpriseManager Designers
Operator	AttunityEnterpriseManager Operators
Viewer	AttunityEnterpriseManager Viewer

Effective permissions are the permissions that take effect when a user is part of more than one group, or when there is a conflict between the user's permission and the group's permission, or in the hierarchy. For details, see [Inheritance and overrides \(page 301\)](#).

Encrypting the User Permissions File

User permissions are stored in the following repository file:

```
<product_dir>\Data\cfgrepo.sqlite
```

To prevent unauthorized access of this file, you can encrypt it using the procedure described in [Replacing the Master User Password \(page 45\)](#).

Granular access control

For each user, Enterprise Manager lets you set granular access permissions for different hierarchy levels in the system and for different objects at the same hierarchy level. This granular access control facilitates decentralization of control, effectively preventing the same user from, for example, accessing endpoints and defining and running tasks. As such, granular access control lets you create a buffer between those who can create and access endpoints (DBAs) and those who can create and run tasks.

Enterprise Manager handles permission management as follows:

- Admins can add, remove, and change permissions.
- Designers and Operators can view permissions.
- Viewers cannot view permissions.

By default, each object inherits its permissions from its parent. The following hierarchy is in place, where:

- **Enterprise Manager Root** refers to all Enterprise Manager server settings and all Replicate servers monitored by Enterprise Manager.



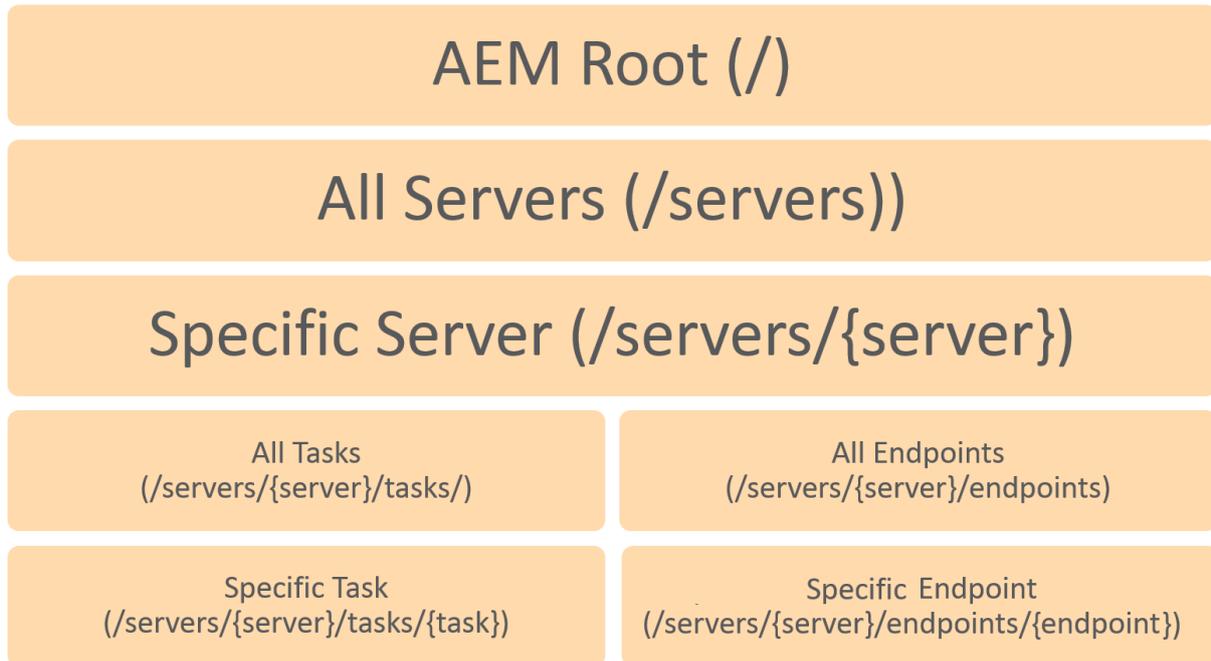
Changes to Enterprise Manager root permissions will affect all levels that inherit those permissions.

- **All Servers** refers to all Replicate servers monitored by Enterprise Manager. This level does not have access to Enterprise Manager server settings.
 - **Specific Server** refers to a server monitored by Enterprise Manager and all its child objects (server settings, tasks, and endpoints).
- **All Tasks** refers to all tasks that run on a specific Replicate server.
 - **Specific Task** refers to all parameters of a particular task.



*To make a user a designer on a task, the user must be at least a viewer on **All Endpoints**.*

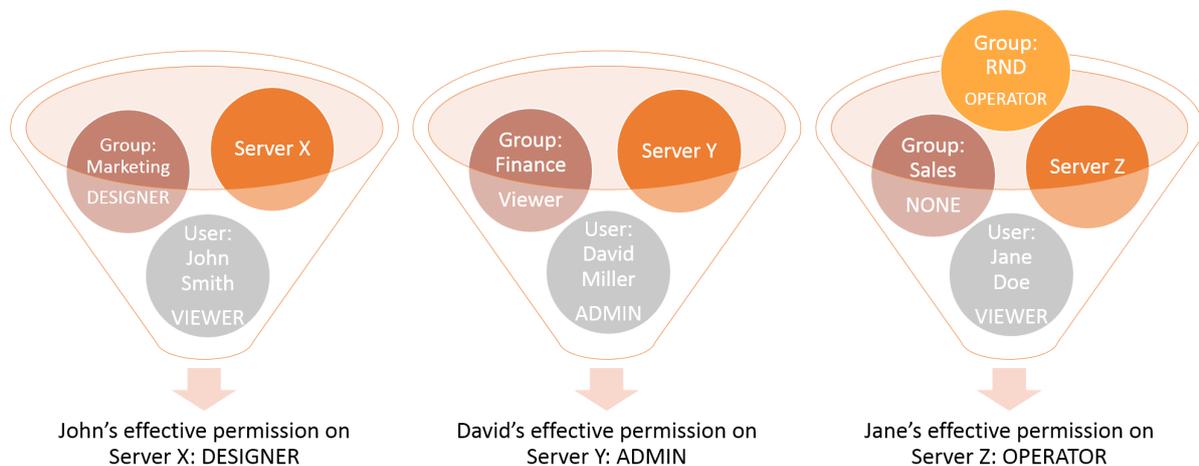
- **All Endpoints** refers to all endpoints connected to a specific Replicate server.
 - **Specific Endpoint** refers to all parameters of a particular endpoint.



Inheritance and overrides

Group permission may contradict the permission that a particular user was granted. In this case, the higher permission overrides the lower permission, as illustrated in the following figure:

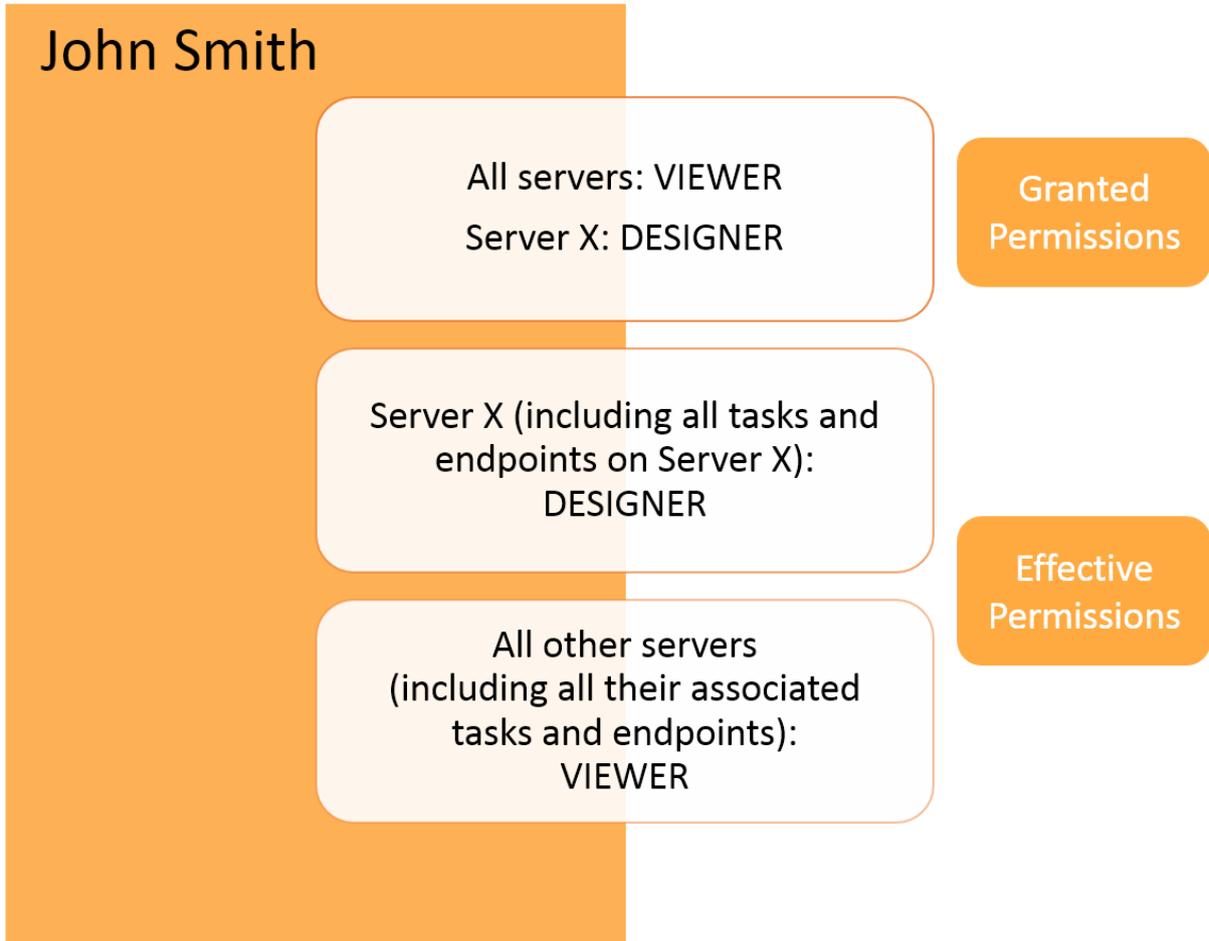
Group vs. user permissions



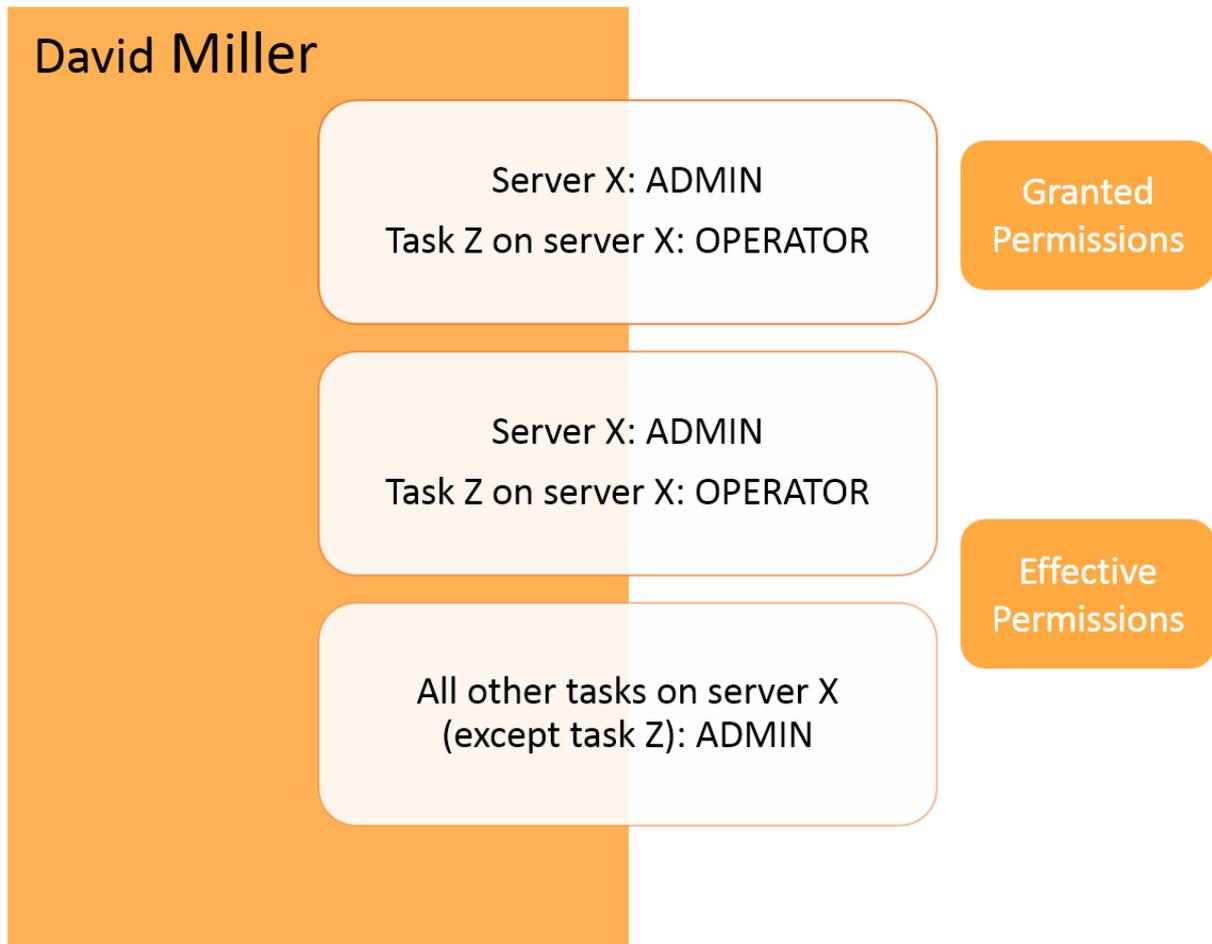
By default, the permission of a user or group object is inherited from the access control list (ACL) of the object's parent. However, a lower or higher permission may override this permission. In this case, the overriding higher permission is the effective permission for the object, stopping inheritance from the parent. As a result, any changes to the parent no longer affect this user or group.

The following figures illustrate these concepts:

Inheritance overridden by higher permission



Inheritance overridden by lower permission



In the **User Permissions** window, inheritance is indicated by a checkmark in the **Inherited** column. By default, inheritance is enabled for all users and groups on any level. Changing permissions by using the slider automatically stops inheritance for the selected user or group. Enterprise Manager also lets you disable inheritance by disconnecting the entire authorization level from the parent level. For information on how to do this, see [Managing user permissions \(page 307\)](#).

Override exceptions

When a user or group is granted any permission higher than None on an object (except an endpoint), it automatically receives Viewer permission on all parent objects in the hierarchy. However, the Viewer permission granted to the parent objects does not enable the user or group to see items in the hierarchy that they were not permitted to see before they received Viewer permission. For clarification, see the example in the following table:

Permission override example

-	Granted Permission	Effective Permission
Server X	None	Viewer
All Tasks on Server X	None	Viewer (limited to Task Z)
Task Z on Server X	Designer	Designer

Roles and permissions

This topic explains how user roles affect the availability of console elements as well as which roles are required in order to perform Enterprise Manager operations.

Availability of console elements according to role

In the Enterprise Manager Console, you see menu items and buttons based on your particular security role. For example:

- The **Servers** view is available to all roles, but Designers only have read access to user permissions, and operators cannot add servers and can only view the different settings, but not edit them. Viewers do not see the options to view logs, edit settings, add, edit, or delete a server, register a license, or start/stop monitoring.
- The **Server** dialog box is available to Admins, Designers, and Operators. Operators can test the connection, but they cannot edit any fields. Viewers do not have access to this dialog box.
- In the **Tasks** view, Operators see the **Open**, **Run**, and **Stop** options, but Viewers only see **Open** option. Operators can search and assign tags, but they cannot add new tags or delete tags.
- In the dedicated **Task** tab, Operators see all available options (**Run**, **Stop**, **Reload**, **Resume**, **Reset data errors**, and so on) as well as the **Monitor** tab, but they do not see the **Designer** tab. Viewers only see the **Monitor** tab. They do not have access to any actions.
- In the **Message Center**, Viewers do not see the option to view logs.

Roles required for Enterprise Manager operations

The following table lists which user role is required to perform the available Enterprise Manager operations.



Some of the task operations are not available/relevant for Qlik Compose tasks. To find out if a particular permission applies to Qlik Compose, refer to the Help topic explaining how to perform the associated operation.

Permissions defined in Enterprise Manager take precedence over the permissions required for performing the corresponding operation directly in Replicate or in Qlik Compose.

Server operation roles

Permission/Operation	Admin	Designer	Operator	Viewer
Servers view	Yes	Read-Only	Read-Only	Read-Only
Add and delete server	Yes	No	No	No
View server connection properties	Yes	Yes	Yes	No
Edit server connection properties	Yes	Read-Only	Read-Only	No
Test server connection	Yes	Yes	Yes	No
Edit column settings in the server list, search for servers, and access the context menu for a specific server.	Yes	Yes	Yes	Yes

Task operation roles

Permission/Operation	Admin	Designer	Operator	Viewer
Access Design view (Replicate only)	Yes	Yes	No	No
Access Monitor view (Replicate only)	Yes	Yes	Yes	Yes
Add and design tasks	Yes	Yes	No	No
Add and edit endpoints	Yes	Yes	No	No
Import Task When a task with the same name already exists on the target server.	Yes	Yes* (see note below)	No	No
Import Task When a task with the same name does not exist on the target server.	Yes	Yes* (see note below)	No	No
Export task without endpoints	Yes	Yes	Yes	No
Export task with endpoints	Yes	Yes	Yes* (see note below)	No



For both of the Import Task permissions mentioned above, if the the exported JSON includes endpoints, then the Enterprise Manager user must also have the role of **Designer** on **All Endpoints** on the target server.



For the 'Export Task with endpoints' permissions mentioned above, the Enterprise Manager user must have the role of **Operator** on the task as well as on both endpoints.

Other operation roles

Permission/Operation	Admin	Designer	Operator	Viewer
Set logging levels, set log file cleanup/rollover, and edit Message Center settings	Yes	Yes	Read-Only	No
Edit user permissions	Yes	Read-Only	Read-Only	No
View logs	Yes	Yes	Yes	No
Perform runtime operations (such as start, stop, or reload targets)	Yes	Yes	Yes	No
Delete tasks	Yes	Yes	No	No
Search for and assign tags	Yes	Yes	Yes	No

Permission/Operation	Admin	Designer	Operator	Viewer
Add and delete tags	Yes	Yes	No	No
Perform Tools menu actions (in Monitor view)	Yes	Yes	Yes Note: The Operator must be for a specific task.	No
Access Message Center	Yes	Yes	Yes	Yes
Register license	Yes	No	No	No
View licenses	Yes	Yes	Yes	No
View the Enterprise Manager machine name in the Licenses tab	Yes	Yes	Yes	No
Hide the main Analytics tab	Yes	Yes	No	No
View license alerts	Yes	Yes	Yes	Yes

Working with local groups

By default, Enterprise Manager only supports working with domain users and groups. To set up support for local groups, you must update the `aemctl.exe.config` file, which is located in the `bin` folder of your Enterprise Manager installation directory (by default, this is `C:\Program Files\Attunity\Enterprise Manager\bin`).

To set up support for local groups:

1. Stop the Qlik Enterprise Manager service.
2. Open the following file in a text editor: `installation directory\bin\aemctl.exe.config`
3. Under `<appSettings>`, add the following row:
`<add key="UseLocalGroups" value="true"/>`

Example:

```
<appSettings>
<!-- LogOverwrite: comma seperated list of loggers ot '*' for enabling global debug -->
<!-- Loggers: Root,Service,Repository,Host,Command,Security,WebLog-->
<add key="LogDebugOverwrite" value=""/>
<add key="AssemblyList" value="AemGlobals,RepuigGlobals"/>
<add key="ClientSettingsProvider.ServiceUri" value=""/>
<add key="DisableToken" value="false"/>
</appSettings>
```

4. Save and close the file.
5. Restart the service.

Managing user permissions

This section explains how to edit user permissions, add and remove users or groups, disable or enable inheritance, restore inherited permissions if they were overridden, and view effective permissions for a user.

By default, inheritance is enabled for all objects (users and groups). This means that permissions are automatically carried over from the parent object. You can turn inheritance on or off for all objects at the current level.

Effective permissions are the permissions that are in effect for a user at any particular level.

For more information on the underlying concepts, see [Granular access control \(page 300\)](#) and [Inheritance and overrides \(page 301\)](#).

To access user permissions at the Enterprise Manager, Analytics, Notifications or All Servers level:

1. Click the User Permissions icon  in the top right corner.
2. In the **Enterprise Manager User Permissions** window, select one of the following tabs:
 - **Enterprise Manager** to specify Enterprise Manager-wide user permissions.



Note Changes to Enterprise Manager permissions will affect all levels that inherit those permissions.

- **Analytics** to specify Analytics permissions.
- **Notifications** to specify notification permissions.
- **All Servers** to specify permissions for all monitored servers.

To access user permissions for a specific Server, All Tasks, or All Endpoints:

1. In Servers view, select the desired server and then select **Permissions** from the **Server Management** toolbar drop-down menu.
2. In the **User Permissions for server: '{server name}'** window, select one of the following tabs:
 - **Server** to specify server-wide user permissions.
 - **All Tasks** to specify permissions for all tasks on this server.
 - **All Endpoints** to specify permissions for all endpoints on this server.

To access user permissions for a specific Task:

- In the tab for a specific task, click the task permissions icon  in the task toolbar. The **User Permissions for task: {task name}** window opens.

To disable inheritance:

1. In the **User Permissions** dialog, click **Disable Inheritance**.



This option disconnects the entire authorization level from the parent level.

2. In the **Disable Inheritance** dialog box, select whether you want to:

- **Convert inherited permissions on this object into explicit permissions:** This option changes inherited permissions to explicit permissions. Any new users or groups will not inherit permissions from the parent.
 - **Remove all inherited permissions from this object:** This option removes all existing permissions inherited from the parent level. Any new users or groups will not inherit permissions from the parent.
3. Click **Disable**.
If you chose to convert inherited permissions, the check mark in the Inherited column changes into an X. If you chose to remove inherited, all users and groups disappear from the list.
 4. Click **OK** to accept the changes or **Cancel** to undo them.

To enable inheritance:

1. In the **User Permissions** dialog, click **Enable Inheritance**.



This option enables inheritance for all users and groups on this level.

2. In the **Enable Inheritance** dialog box, select whether you want to:
 - **Inherit all permissions from parent and override any definition manually made at this level:** This option reinstates inherited permissions for all users and groups that are already defined, and new users and groups will inherit their permissions from the parent level.
 - **Inherit all permissions from parent but keep definitions manually made at this level:** This option preserves the permissions already defined for the existing users and groups and adds all permissions from the parent level. New users and groups will inherit permissions from the parent level.
3. Click **Enable**.
4. Click **Save** or **OK** to accept the changes, or **Discard Changes** or **Cancel** to undo them.

To edit user permissions:

1. In the **User Permissions**, adjust the permission slider for a user or group as desired.



Adjusting the slider stops inheritance from the parent object.

2. Click **OK** to accept the changes or **Cancel** to undo them.

To add a user or group:

1. In the **User Permissions** dialog, click **Add**.
2. In the **Add User/Group** dialog box, select **User** or **Group**.
3. Enter the name for the new user or group in the following format:
 - NetBIOS_name\user (for example: qa\qa)
 - machine_name/local_user (for example: re2008r2js1\JohnMill)
 - *username* - This format is supported with [SAML authentication only](#). The user/group name can contain any Unicode character up to 255 characters and must be a valid Identity Provider user (Okta or Microsoft Azure).
4. Click **OK** to add the user or group and close the dialog box.

5. Click **OK** to accept the changes, or **Cancel** to undo them.

To remove a user or group:

1. In the **User Permissions** dialog, select the user or group you want to remove.
2. Click **Remove**.
3. When prompted, click **Yes** to confirm.
4. Click **OK** to accept the changes, or **Cancel** to undo them.

To restore inherited permissions for a single user or group if they were overridden:

1. In the **User Permissions** dialog, select the user or group.
2. Click **Restore Inheritance** .

The check mark returns to the Inherited column to indicate that permissions for this user or group are inherited from the parent.

To view effective permissions for a user:

1. In the **User Permissions** dialog, do one of the following:
 - Select a user in the list on the left.
 - If a user does not appear in the list but exists in the system and is part of a group, enter the user name in the text field in the **Effective Permissions** pane on the right. Make sure to use the following format:
 - NetBIOS_name\user (for example: qa\qa)
 - machine_name/local_user (for example: re2008r2js1\JohnMill)
2. Click **Get Effective Permissions**.

The effective permissions for the user you entered appear below the button.

Managing Personal Access Tokens

In the Personal Access Tokens tab, you can view the status of all Personal Access Tokens in the system and revoke a user's Personal Access Token. Note that if you revoke a user's Personal Access Token, it will no longer be possible to use the Enterprise Manager API to log in with that token.



*The **Personal Access Tokens** tab is only visible to Enterprise Manager admins.*

To revoke a user's token

1. Select the user and click **Revoke**.
2. When prompted to confirm the operation, click **Yes**.

12.3 Creating an audit trail

An audit trail provides you with information about the transactions executed in Enterprise Manager. Enterprise Manager traces operations that require a minimum role of operator. For these operations, the audit trail shows who did what, when, and on which objects. By default, the audit trail is enabled. Audit trail files are located in the following folder: `<Enterprise Manager installation folder>\data\AuditTrail\audit_service`.



The audit trail is secure. Audit files are compressed and protected by checksum.



Do not edit or delete the audit files. Any changes to these files might cause the system to crash.

Enterprise Manager retains audit files for two weeks or until they reach a total size of 100 MB (10 files). You can configure these settings through the command line interface (CLI). For more information, see [Configuring Enterprise Manager using the CLI \(page 313\)](#), in particular the [aemctl.exe audit_trail control](#) command.

In addition, you can manually download an audit trail file in .csv format. This file includes filtered actions for a specific time frame or a custom time range.

Audit trail files may contain all or some of the following information, depending on whether the object exists in Enterprise Manager:

- Timestamp (UTC)
- User
- Node
- Requested Action
- Required Permission
- Effective Permission
- Security Result
- Action Result
- Error Message
- Server
- Task
- Endpoint
- Notification
- Enable
- Delete_task_logs
- Tag
- Schema
- Table
- Delete_fts_logs
- FTS (File Transfer Service)
- Payload

To view payload information, you can copy the link in the **Payload** column and paste it into the address bar of a browser window.

For information on decoding stream payloads, see [Decoding an encoded stream payload \(page 311\)](#).



The REST information listed in the following table is not presented in the audit trail columns. It is only accessible via the payload.

REST access points

REST	Info Access From
puterrorbehavior	payload - task name
tablecontrol	payload - task name
test_database_async	payload - endpoint name
browse_connection_async	payload - endpoint name
action=test_async_complete	payload - endpoint name
action=browse_connection_async	payload - endpoint name

To manually download an audit trail file:

1. Click the Audit Trail icon  in the top right corner.
2. In the **Audit Trail** dialog box, from the **Time Range** list, select a time range.
3. If you select **Custom**, also specify the **From** date and time and the **To** date and time. The time you select is the local machine time. This gets converted to coordinated universal time (UTC) in the resulting CSV file.
4. Click **Generate**. The *AuditTrail.csv* file is created and downloaded. Depending on your browser, you should see it at the bottom of your browser window or in a separate window.

Decoding an encoded stream payload

Some audit records (e.g. RegisterLicenses) may contain an encoded stream payload. Encoded payloads are displayed as byte arrays and need to be decoded using Base64.

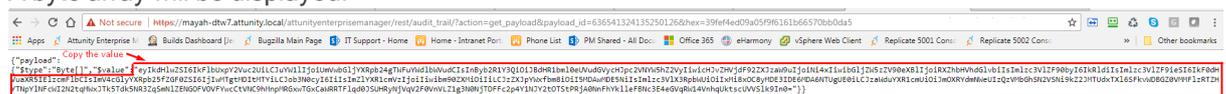
To decode an encoded stream payload:

1. Locate the payload URL in the audit record.

H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X	Y	Z	AA	AB	AC	AD	AE	AF	AG	AH	AI	AJ
Action Res	Error SE	Er	MC	SI	T	D	E	P	ayload																			
success																												
success																												

2. Copy the URL into your browser's address bar and press [Enter].

A byte array will be displayed.



- Copy the byte array into a Base64 decoder and decode it.

BASE64 Decode and Encode
 Have to deal with Base64 format? Then this site is made for you! Use the super simple tool. If you are interested about the inner workings of the Base64 format, just read the detailed description.

Decode
Encode
Other tools

Decode from Base64 format

Simply use the form below

```
"eyllkdHlwZSI6IkFibUxpY2Vuc2UiLCJuYW1lIjoiaUUmVwbGijYXRpb24gTWFuYWdlbWVudClnByb2R1Y3QiOiJBdHR1bml0eUJvudGVycHJpc2VNYW5hZ2VyliwicHJvZHVjZD92ZXJzaW9uIjoiaU4xliwibGijZW5zZV90eXBlljoiaXRhbHVhdGlvbiImlzc3VlZF90byI6IkRldiImlzc3VlZF9ieSI6IkF0dHVuaXR5IElzcmluZmV4cGlyYXRpb25fZGF0ZSI6IjwMTgtMDItMTYiLCJkaWVudCI6ImVudCmVlZjoiIiwibm90ZXMiOiIiLCJzZXJpYWxibm8iOiIiLDE5MDAwMDE5NiImlzc3VlZ3RpbWUiOiIiLCJmVlZjoiIiwibm90ZXMiOiIiLCJzaWduYXR1cmUiOiIiLCJmOXRydmNWwvX0FgELDYsE6Ga3ibSEqb67kj1IINNsvNMGvjJcedCF8UNTv0p+U4/a2zLDlpLiBidQlyjwBRPTr65jWatVuKgX77CcL1E5jxcSickNM+OF046qablexPMsq8xejEmxVxjRklqEUJY="
```

< DECODE >
UTF-8
You may also select input charset.

Live mode OFF
Decodes while you type or paste (strict format).

Note that decoding of binary data (like images, documents, etc.) does not work in live mode.

UPLOAD FILE
Decodes an entire file (max. 10MB).

With an array of smart features, Freshdesk™ lets you support almost instantly. Sign up now

```
{
  "Type": "AemLicense",
  "name": "Replication Management",
  "product": "AttunityEnterpriseManager",
  "product_version": "6.1",
  "license_type": "Evaluation",
  "issued_to": "Dev",
  "issued_by": "Attunity Israel",
  "expiration_date": "2018-02-16",
  "hosts": "",
  "features": "",
  "notes": "",
  "serial_no": "90000196",
  "issue_time": "12/18/2017 1:00:55 PM",
  "signature": "f9tXvcVyB3C5Llhr7eR6/dgbLMGqMyzHY/X0FgELDYsE6Ga3ibSEqb67kj1IINNsvNMGvjJcedCF8UNTv0p+U4/a2zLDlpLiBidQlyjwBRPTr65jWatVuKgX77CcL1E5jxcSickNM+OF046qablexPMsq8xejEmxVxjRklqEUJY="
}
```

13 Configuring Enterprise Manager using the CLI

You can use the Enterprise Manager command line interface (CLI) to modify the data directory, change the default configuration settings, and manage the Qlik Enterprise Manager service during installation or when configuring Enterprise Manager to run on a cluster.

All commands should be run from `<PRODUCT_INSTALLATION_DIRECTORY>\bin`

To get help when using the command line, you can run the Help command. For example, for help about the parameters available with the service command, run the following command (shown with the default installation path):

```
C:\Program Files\Attunity\Enterprise Manager\bin>aemctl.exe service help
```

This brings up the list of help parameters.

Or, for a list of the available commands, run:

```
aemctl.exe help
```



*When the Enterprise Manager data directory is installed in a non-default location, the `-d <data_directory>` parameter must be included in all commands, where `data_directory` is the location of the **data** directory.*

Running CLI commands requires the admin role. For more information, see [User permissions \(page 298\)](#).

The login authentication method is also set using the CLI. For more information, see [Setting the login authentication method \(page 25\)](#).

13.1 Setting the Enterprise Manager host address

To set the host address, run the following command:

```
aemctl.exe configuration set --address hostAddress
```

where `hostAddress` is the address of the Enterprise Manager server machine.



When using a Load Balancer, `hostAddress` should be the Load Balancer host address.

Abbreviated parameter: `-a`

13.2 Setting the Enterprise Manager HTTP port

To set the HTTP port, run the following command:

```
aemctl.exe configuration set --http_port port
```

Abbreviated parameter: `-p`

Default: 80

13.3 Setting the Enterprise Manager HTTPS port

To set the HTTPS port, run the following command:

```
aemctl.exe configuration set --https_port port
```

Abbreviated parameter: -s

Default: 443

13.4 Setting the Enterprise Manager root URL

To set the root URL, run the following command:

```
aemctl.exe configuration set --root_url url
```

Abbreviated parameter: -r

13.5 Showing the Enterprise Manager version

To show the version, run the following command:

```
aemctl.exe configuration set --version
```

13.6 Showing the Enterprise Manager CLI Help

To show the Help, run the following command:

```
aemctl.exe configuration set --help
```

13.7 Service Commands

To stop the Qlik Enterprise Manager service, run the following command:

```
aemctl.exe service stop --name servicename [--timeout seconds]
```

Where:

servicename is the name of the Enterprise Manager service

seconds is the time to wait in seconds before stopping the service

To start the Qlik Enterprise Manager service, run the following command:

```
aemctl.exe service start --name servicename [--timeout seconds]
```

Where:

servicename is the name of the Enterprise Manager service

13 Configuring Enterprise Manager using the CLI

`seconds` is the time to wait in seconds before starting the service

To bring up a list of other service-related parameters, run the following command:

```
aemctl.exe service
```

13.8 Cleaning the self-signed certificate

To clean (i.e. clear) the self-signed certificate, run the following command:

```
aemctl.exe certificate clean
```

13.9 Setting the audit trail retention size and age

To see the audit trail size and/or age limit, run the following command:

```
aemctl.exe audit_trail control [--age hours] [--size megabytes]
```

Where:

`hours` is the number of hours to retain the audit trail file. The default is 168 hours, which is one week.

`megabytes` is the maximum size of the audit file to retain. The default is 500 MB.

13.10 Master User Password commands

The following section describes commands that can be used to generate a Master User Password, set the Master User Key, and change the Master User Key.

For more information on the role of the master user password and the master user key, see [Replacing the Master User Password \(page 45\)](#).

Generating a random Master User Password

To generate a random master user password, run the following command:

```
aemctl.exe utils genpassword
```

Setting or changing the MUK (Master User Key)

Run this command to set the Enterprise Manager MUK (Master User Key) as part of the corporate security policy or when configuring Enterprise Manager to work in a cluster.

For information on installing Enterprise Manager on a cluster, see [Installing Qlik Enterprise Manager in a Windows cluster \(page 347\)](#)



The password must be at least 32 characters.

13 Configuring Enterprise Manager using the CLI

If you add the `--prompt` parameter to the command and omit the `-p` and `-c` parameters, the CLI will prompt you for the password. When you enter the password, it will be obfuscated. This is especially useful if you do not want passwords to be retained in the command prompt history.

Syntax:

```
aemctl.exe [-d data_directory] masterkey set --prompt
```

To set the MUK, run the following command:

```
aemctl.exe [--data data_directory] masterkey set --password password
```

Where:

`--data` (or `-d` in its abbreviated form) is used to specify the location of the Enterprise Manager **data** folder, but is only required if the **data** folder is not on the same drive as the **bin** folder.

To change the MUK, run the following command:

```
aemctl.exe [--data data_directory] masterkey set --password new_password --current-password old_password
```

Where:

`--data` (or `-d` in its abbreviated form) is used to specify the location of the Enterprise Manager **data** folder, but is only required if the **data** folder is not on the same drive as the **bin** folder.

Setting or changing the Java MUK (Master User Key)

Run this command to set the Enterprise Manager MUK (Master User Key) as part of the corporate security policy or when configuring Enterprise Manager to work in a cluster.

For information on installing Enterprise Manager on a cluster, see [Installing Qlik Enterprise Manager in a Windows cluster \(page 347\)](#)



The password must be at least 32 characters.

To set the Java MUK, run the following command:

```
atajs.bat [--data JavaDataFolderFullPath] masterkey set password
```

Where:

`--data` (or `-d` in its abbreviated form) is used to specify the location of the Enterprise Manager **data** folder, but is only required if the **data** folder is not on the same drive as the **bin** folder.

To change the Java MUK, run the following command:

```
atajs.bat [--data JavaDataFolderFullPath] masterkey set new_password old_password
```

Where:

13 Configuring Enterprise Manager using the CLI

- `--data` (or `-d` in its abbreviated form) is used to specify the location of the Enterprise Manager **data** folder, but is only required if the **data** folder is not on the same drive as the **bin** folder.
- `new_password` is the new password.
- `old_password` is the current password.

13.11 Showing the connectivity and login settings

To show the current connectivity and login settings, run the following command:

```
aemctl configuration show
```

The output will be as shown below.

Connectivity settings

- `-a, --address` - The Enterprise Manager host address.
- `-p, --http_port` - The HTTP port through which Enterprise Manager is accessed.
- `-s, --https_port` - The HTTPS port through which Enterprise Manager is accessed.
- `-r, --root_url` - The Enterprise Manager URL root.
- `-u, --user_timeout` - The user idle timeout (minutes).
- `-m, --domain` - The Enterprise Manager domain name.
- `--authentication_method` - FORM, SSO or SAML.

SAML settings



If SAML was configured in the past, these parameters will always be shown (even if SAML is not the current authentication method).

- `--idp_url` - The SAML IdP URL.
- `--idp_issuer` - The unique identity of the SAML IdP.
- `--idp_certificate_file` - A file containing the certificate from the SAML IdP.
- `--idp_username_attribute` - The user name specified in the SAML assertion document.
- `--idp_user_displayname_attribute` - The user display name specified in the SAML assertion document.
- `--idp_user_groups_attribute` - The group name specified in the SAML assertion document.

For information on setting up SAML and other types of authentication, see [Setting the login authentication method \(page 25\)](#).

13.12 Fine tuning performance

This section describes various parameters that you can set to tweak performance.

Turning off the Analytics Server

If you have configured the [Analytics module](#) and you are running multiple instances of Enterprise Manager, for each Enterprise Manager instance there will be a corresponding Analytics Server instance. As only one Analytics Server instance is required, you can turn off the other instances to improve performance.

To do this:

1. Run the following command:
`aemctl analytics set --noserver`
2. Restart the Qlik Enterprise Manager service.

To turn the Analytics Server back on:

1. Run the following command:
`aemctl analytics set --runserver`
2. Restart the Qlik Enterprise Manager service.

Changing the update intervals

If you are encountering performance issues due to a large number of monitored tasks or servers (Replicate or Compose), increasing the update intervals should significantly improve performance.

Changing the server update interval

The server update interval determines how often Enterprise Manager queries all of the monitored servers for updated statistics on tasks, views, and tags.

To change the interval, run the following command (replacing *your-interval* with the desired interval in seconds):

```
aemctl manager set --server_update_interval your-interval
```

Abbreviated parameter: `-u`

Default: 5 seconds

Changing the web console refresh interval for servers

The web console update interval for servers determines how often the browser queries Enterprise Manager in order to refresh the associated pages.

To change the interval, run the following command (replacing *your-interval* with the desired interval in seconds):

```
aemctl manager set --web_update_server_interval your-interval
```

Abbreviated parameter: `-s`

Default: 3 seconds

Changing the web console refresh interval for tasks

The web console update interval for tasks determines how often the browser queries Enterprise Manager in order to refresh the associated pages.

13 Configuring Enterprise Manager using the CLI

To change the interval, run the following command (replacing *your-interval* with the desired interval in seconds):

```
aemctl manager set --web_update_task_interval your-interval
```

Abbreviated parameter: `-t`

Default: 5 seconds

Changing the data tree cache interval

The data tree cache interval determines how often a cache of Enterprise Manager's data is created. When set, the web console will read from the cached copy instead of the main data tree according to the specified interval. Usually, this parameter does not need to be changed. However, if changing the other performance parameters does not resolve your performance issues, adjusting this parameter might help.



The minimum interval that can be set is five seconds.

To change the interval, run the following command (replacing *your-interval* with the desired interval in seconds):

```
aemctl manager set --data_tree_cache_interval your-interval
```

Abbreviated parameter: `-c`



*To turn off data tree caching, set *your-interval* to `-1`*

Default: Off

14 Cataloging tasks in Qlik Catalog

Leveraging Qlik Catalog's powerful cataloging capabilities allows you to gain insights into your data assets and take appropriate action. You can select any Replicate task in Enterprise Manager and catalog all of the target table metadata.

In this section:

- [Terminology \(page 320\)](#)
- [Prerequisites \(page 321\)](#)
- [Setting up connectivity to Qlik Catalog \(page 321\)](#)
- [Limitations and considerations \(page 321\)](#)
- [Catalog operations \(page 322\)](#)

14.1 Terminology

Leveraging Qlik Catalog's powerful cataloging capabilities allows you to gain insights into your data assets and take appropriate action. You can select any Replicate task in Enterprise Manager and catalog all of the target table metadata.

Replicate users may find some of the Qlik Catalog terminology a little confusing at first; for instance, the Replicate target endpoint is referred to as the Source Connection in Qlik Catalog. To lessen the potential for confusion, some of the cataloging terms used in Enterprise Manager differ from those used in Qlik Catalog. While this may seem counter-intuitive at first, it is hoped that using terms that are appropriate within the specific context of each application will actually create a far more intuitive user experience.

Before getting started then, it's important to understand how the cataloging and replication terminology used in Enterprise Manager is reflected in Qlik Catalog.

The table below lists the terms in use by Enterprise Manager and the equivalent term in Qlik Catalog.

Equivalent terms and values

Enterprise Manager Term	Catalog Term	Enterprise Manager Default Value
N/A	Source	<ReplicateServer>_<TargetEndpoint>_<Schema>
Datastore	Source Connection	<ReplicateServer>_<TargetEndpoint>
Location	Source Hierarchy	Replicate.<ReplicateServer>_<ReplicateTask>
Data Asset	Datasets	<ReplicateServer>_<ReplicateTask>
Catalog AD Group	Group	<The Active Directory group defined in the Qlik Catalog connection settings >
Column	Field	N/A
Table	Entity	N/A

14.2 Prerequisites

Before you begin to work with Qlik Catalog, make sure the following prerequisites have been met:

- Replicate 7.0 or later installed in your network
- Make a note of your Qlik Catalog [access information](#) (hostname, username, password, etc.). You will need it to configure connectivity to the Qlik Catalog Server.

14.3 Setting up connectivity to Qlik Catalog

To catalog Replicate tasks in Qlik Catalog, Enterprise Manager needs to establish and maintain a connection with the Qlik Catalog server.

To provide your Qlik Catalog connection details:

1. Open the **Settings** window and select **Qlik Catalog Server Connection** on the left.
2. Enter the following information:
 - **Host:** The host name of the Qlik Catalog server machine.
 - **Port:** The port over which the connection will be established. The default port is 8443.
 - **User name:** The user name for connecting to the Qlik Catalog server machine.
 - **Password:** The password for connecting to the Qlik Catalog server machine.
 - **Web app name:** `qdc`
 - **URL:** The URL link is created automatically from the host name, the port and the Web app name.
 - **Default catalog AD group:** The Active Directory group in Qlik Catalog under which tasks will be cataloged. This will be the default group [when cataloging tasks](#).
3. Optionally, click **Test Connection** to verify your connection settings.
4. Click **Save** to save your setting or **Discard Changes** to revert to your previous settings.

Catalog columns

After you configure connectivity to a Qlik Catalog server, the following columns become visible in Tasks View.

- **Cataloged** - Indicates whether or not the task is cataloged.
- **Name** - The task's data asset (dataset) name in Qlik Catalog.
- **Location** - The task's location in the Qlik Catalog hierarchy.
- **Datastore** - The task's datastore (source connection) name in Qlik Catalog.
- **AD Group** - The Active Directory group in Qlik Catalog under which the task is cataloged.

14.4 Limitations and considerations

Working with Qlik Catalog is subject to the following limitations:

- Compose tasks are not supported.
- Cataloging a task with a large number of source tables might take some time.

- Replicate streaming target endpoints (such as Kafka) are not supported. For a full list of streaming endpoints, see the Replicate Support Matrix.
- Replicate file-based target endpoints (e.g. Amazon S3) are not supported.
- Operations performed directly in Qlik Catalog on cataloged Replicate tasks and sources will not be synced back to Enterprise Manager. For example, if you delete a source in Qlik Catalog, the task will still appear as cataloged in Enterprise Manager.
- Tasks that are deleted in Enterprise Manager when a server is not being monitored will not be deleted in Qlik Catalog when monitoring is resumed for that server. Such tasks will need to be manually deleted in Catalog.
- After upgrading to Enterprise Manager 7.0, existing tasks need to be stopped and then resumed in order to be eligible for cataloging.
- Data cannot currently be ingested from the Replicate target tables (entities) into Qlik Catalog.

14.5 Catalog operations

You can catalog, recatalog, and uncatalog tasks. This topic describes how and when such operations should be performed.

Cataloging tasks

To catalog a task:

1. In Tasks view, select the task you want to catalog and then click the **Catalog** toolbar button and select **Catalog**.

A window displaying the following auto-generated properties opens:

- **Data asset name in catalog:** This is a combination of the Replicate server and task name. So, for example, if the server name is *RepServer* and the task name is *SQLtoAWS*, the data asset name will be *RepServer_SQLtoAWS*.
- **Catalog data asset under:** By default, all data assets are cataloged under *Replicate*. For example, a data asset named *RepServer_SQLtoAWS* would appear as *Replicate.RepServer_SQLtoAWS*. You can create additional hierarchies (or sub-locations using Enterprise Manager terminology), using dots as separators. For example, to add a top-level container called *QDI* to *Replicate.RepServer_SQLtoAWS*, you would need to enter the following:
QDI.Replicate.RepServer_SQLtoAWS
- **Datastore in catalog:** This is a combination of the Replicate server name and the target endpoint name.
- **Catalog AD group:** The name of the Active Directory Group defined in Qlik Catalog under which the task will be cataloged.

You can edit these properties as required.

2. Click **Catalog**.

If the cataloging operation is successful, a confirmation will be displayed at the top of the page and a new **Catalog** tab with the cataloging properties (described in step 1 above) will be added to the monitoring dashboard in the right pane.

Cataloging example

In the following example, a Replicate task has copied the following tables (shown as schema.**table**) from a PostgreSQL on-premises database to a PostgreSQL in the cloud (AWS): `HumanResources.Department`, `HumanResources.Employee`, `Purchasing.ShipMethod`, `Purchasing.Vendor`, `Sales.Customer`, and `Sales.Store`. The customer needs to ensure the validity and integrity of these tables for the purpose of analytics. To this end, the customer has decided to catalog the Replicate task.

After cataloging the Replicate task, the associated target tables immediately become visible in Qlik Catalog:

The screenshot shows the Qlik Catalog interface with a sidebar on the left containing navigation options: Catalog, Ops, Source, Discover, Prepare, Publish, and Security. The main area displays a grid of six table cards, each representing a cataloged table. Each card includes a search icon, a source identifier, and three circular gauges for Operational, Quality, and Popularity metrics. Below the gauges, the 'Entity Level' is listed as 'ADDRESSED', along with Row Count, Field Count, and Last Load information.

Table Name	Source	Operational	Quality	Popularity	Entity Level	Row Count	Field Count	Last Load
Department	Replicate_184_AW_...	N/A	N/A	0%	ADDRESSED	N/A	4	N/A
Employee	Replicate_184_AW_...	N/A	N/A	0%	ADDRESSED	N/A	16	N/A
ShipMethod	Replicate_184_AW_...	N/A	N/A	0%	ADDRESSED	N/A	6	N/A
Vendor	Replicate_184_AW_...	N/A	N/A	0%	ADDRESSED	N/A	8	N/A
Customer	Replicate_184_AW_...	N/A	N/A	0%	ADDRESSED	N/A	7	N/A
Store	Replicate_184_AW_...	N/A	N/A	0%	ADDRESSED	N/A	6	N/A

Clicking the **Source** tab shows the list of schemas to which the tables belong.

Action	Name	Business Name
More ▾	Replicate_184_AW_Target_EP_Sales	
More ▾	Replicate_184_AW_Target_EP_Purchasing	
More ▾	Replicate_184_AW_Target_EP_HumanResources	

For a detailed description of the tasks that can be performed within Qlik Catalog, refer to the Qlik Catalog Help.

Uncataloging tasks

You can uncatalog a task by simply selecting the desired task, clicking the **Catalog** toolbar button and then selecting **Uncatalog**.



When you delete a cataloged task from Enterprise Manager, the task is automatically removed from Qlik Catalog as well.

Recataloging tasks

Recataloging task uncatalogs and then catalogs a task in a single operation. Recataloging can be used to remedy out-of-sync situations resulting from actions performed directly in Catalog (such as deleting a source).

You can recatalog a task by simply selecting the desired task, clicking the **Catalog** toolbar button and then selecting **Recatalog**.

15 Analytics

Enterprise Manager Analytics can be used to measure server and task metrics over a specific time-period. Examples of server metrics that can be measured include memory consumption and disk usage. Examples of task metrics that can be measured include the number of tables and records processed, throughput, and the number of changes applied.

You can also filter the information according to server, source database, target database, and task type.



Analytics is currently provided for Replicate tasks only.

Analytics information is useful in the following cases:

- Capacity Planning
- Sizing of Replicate Servers
- Replicate Performance Monitoring
- Historical Trend Analysis

In this section:

- [Prerequisites \(page 325\)](#)
- [Permissions \(page 327\)](#)
- [Analytics dashboards \(page 327\)](#)
- [Exporting to TSV \(page 335\)](#)
- [Creating filters \(page 335\)](#)
- [Using the Pivot Menu \(page 337\)](#)
- [Analytics repository schema \(page 337\)](#)

15.1 Prerequisites

This section describes the prerequisites for working with the Enterprise Manager Analytics module.

Install PostgreSQL

Enterprise Manager Analytics data is stored in a PostgreSQL database. Therefore, prior to using Enterprise Manager Analytics, make sure that PostgreSQL 17.4 or later is installed either on the Enterprise Manager machine or on a machine that is accessible from Enterprise Manager.

For your convenience, the PostgreSQL setup file is included with Enterprise Manager. If you would like to install PostgreSQL on the Enterprise Manager machine and did not do so during the Enterprise Manager installation, you can find the PostgreSQL installer in the following location:

```
<Enterprise_Manager_INSTALLATION_FOLDER>\postgres\kit
```

For instructions on installing and maintaining PostgreSQL, refer to the PostgreSQL Help.

Create a dedicated database and assign the required privileges

- Create a dedicated database and user in PostgreSQL that will own the tables accessed by the Enterprise Manager Analytics module.
- Recommended: Create a dedicated tablespace for the Enterprise Manager tables and allocate it as the default tablespace for the Enterprise Manager database.
- The Enterprise Manager user will need ALL PRIVILEGES on the Enterprise Manager database.

Configure connectivity to PostgreSQL

Configure connectivity to the PostgreSQL repository as described in [Repository connection settings \(page 292\)](#).

Set up data collection and purging from PostgreSQL

Configure data collection and purging settings as described in [Analytics - Data collection and purge settings \(page 293\)](#).

Register a license

A Replication Analytics license is required in order to use Analytics. If no license is detected, a **Register License** message will be displayed in the main **Analytics** tab.

If you have a license, you can register it by following the procedure described in [Registering a license \(page 297\)](#).



If you do not intend to use the Analytics feature, you can hide the main **Analytics** tab by clicking the **Hide Analytics Dashboards** link in the **Register License** message.

Obtaining a license



The procedure below does not apply when installing Enterprise Manager in a [High Availability Cluster](#). To obtain a Replication Analytics license for Enterprise Manager in a High Availability Cluster, you must provide your Qlik Sales Representative with the following information, depending on which Windows version the cluster is running:

- **Windows Server 2016:** The official cluster FQDN.
- **Windows Server 2012 R2:** The FQDN of each of the cluster nodes and the official cluster FQDN.

If you do not have a license, you can obtain one as follows:

1. Copy the Enterprise Manager machine name from either of the following locations:
 - The **Register License** message that is displayed in the main **Analytics** tab.
 - The bottom of the **Licenses** tab [in the Settings window](#).

2. Request a license from your Qlik Sales Representative, making sure to include the Enterprise Manager machine name in your request.

Port

Make sure that the PostgreSQL port (5432 by default) is open for outbound communication.

Hardware

If the PostgreSQL database is installed on the Enterprise Manager server and there are less than 1000 tasks, then the [Enterprise Manager hardware guidelines](#) should be enough for the PostgreSQL database as well.

If you wish to install PostgreSQL on a different server, then the following minimum system requirements are recommended:

- 8 GB RAM
- 4 CPU Cores
- 10 GB disk space database tablespace

15.2 Permissions

Analytics has its own set of permissions which are inherited from Enterprise Manager by default.

These are as follows:

- Viewers, Operators and Designers can access the Analytics dashboards and run reports.
- Administrators can access the Analytics dashboards and also access the Analytics settings to configure the PostgreSQL repository, control how much data is stored in the Repository and start/stop the Analytics processes.

15.3 Analytics dashboards

In the **Analytics** tab, you can review server and task trends over a specific time period. For each graph, you can choose whether to display weekly, hourly, daily (the default), monthly, or by minute data. You can also filter the data to show information for a particular timeframe and for particular tasks, Replicate servers, source databases, and target databases.

Dashboards can be accessed either by selecting the desired dashboard from the drop-down list on the left of the toolbar, or by expanding the pivot menu on the right of the **Analytics** tab and clicking the desired dashboard link.

The following dashboards are available:

- [Trends](#)
- [Trends by Server](#)
- [Trends by Task](#)
- [Top Servers](#)

- [Top Tasks](#)
- [Capacity Planning](#)

Trends

The **Trends** dashboard plots key performance and activity metrics for the given time frame. Below each chart, you can toggle to view the trending data by hour, by day, by week, by month, or by minute.

For the specified filters, the Trends dashboard displays the following charts:

- [Server utilization \(page 328\)](#)
- [Full load \(page 328\)](#)
- [Change processing \(page 328\)](#)
- [Errors \(page 329\)](#)

Server utilization

The **Server Utilization** section contains the following graphs:

- **Memory** - Shows the average and maximum memory consumption across all specified servers and tasks.
- **Disk Utilization** - Shows the average and maximum disk space utilization across all specified servers and tasks.
- **Qlik CPU** - Shows the average and maximum Qlik CPU per server (as a percentage) across all specified servers and tasks.
- **Machine CPU** - Shows the average and maximum machine CPU per server (as a percentage) across all specified servers and tasks.
- **Task CPU** - Shows the average and maximum CPU per task (as a percentage of server CPU) across all specified servers and tasks.

Full load

The **Full Load** section contains the following graphs:

- **Throughput** - Shows the average source and target throughput (in rec/sec) per task.
- **Tables** - Shows the total number of completed, loading, queued and error tables across all specified servers and tasks.
- **Records** - Shows the total number of completed and remaining records across all specified servers and tasks.



For the Full Load charts, the data is not aggregated and cannot be viewed by hour, by day, by week, by month, or by minute. Rather, data is displayed for key points in time for the given time frame.

Change processing

The **Change Processing** section contains the following graphs:

- **Applied Changes** - Shows the total number of changes applied, grouped by INSERT, UPDATE, DELETE and DDLs across all specified servers and tasks.
- **Throughput and Latency**- Shows the average source and target throughput (in rec/sec) and the average source and apply latency (in seconds) per task.
- **Change Volume** - Shows the total COMMIT change volume across all servers and tasks.
- **Changes in Memory and Disk** - Shows the average number of changes accumulated in memory and on disk until source commit as well as the average number of changes in memory and on disk during apply, and until target commit per task.

Errors

The graph in the **Errors** sections shows the total number of data errors across all specified servers and tasks, including both Full Load and Change Processing tasks.

Trends by server

The **Trends by Server** dashboard plots key performance and activity metrics for the given time frame, according to server. Trends for the top 10 servers will be displayed based on the selected Rank By metric. If necessary, narrow your filter set to investigate a specific set of servers.

Below each chart, you can toggle to view the trending data by hour, by day, by week, by month, or by minute. For the specified filters, the Trends by Server dashboard displays the following charts:

- [Server utilization \(page 329\)](#)
- [Full load \(page 329\)](#)
- [Change processing \(page 329\)](#)
- [Data errors \(page 330\)](#)

Server utilization

The **Server Utilization** section contains the following graphs:

- **Average Memory** - The average memory consumed by all tasks on a server.
- **Average Disk Utilization** - The average disk space utilization by all tasks on a server.
- **Average Qlik CPU** - The average CPU utilization per server by Replicate and Enterprise Manager processes only.
- **Average Machine CPU** - The average CPU utilization per server by *all* processes (including Replicate and Enterprise Manager processes).
- **Average Task CPU** - The average CPU utilization per task (as a percentage of server CPU) for each server.

Full load

The **Full Load** section contains the following graph:

- **Average Throughput** - The average target throughput (in rec/sec) per task for each server

Change processing

The **Change Processing** section contains the following graphs:

- **Total Applied Changes** - The total number of applied changes for all tasks on a server.
- **Average Target Throughput** - The average target throughput (in rec/sec) per task for each server.
- **Average Apply Latency**- The average apply latency (in seconds) per task for each server.
- **Total Change Volume** - The total COMMIT change volume for all tasks on a server.

Data errors

The total number of data errors across all tasks on a server, including both Full Load and Change Processing tasks.

Trends by tasks

The **Trends by Task** dashboard plots key performance and activity metrics for the given time frame, according to task. Trends for the top 10 tasks will be displayed based on the selected Rank By metric. If necessary, narrow your filter set to investigate a specific set of tasks.

Below each chart, you can toggle to view the trending data by hour, by day, by week, by month, or by minute. For the specified filters, the **Trends by Task** dashboard displays the following charts:

- [Server utilization \(page 330\)](#)
- [Full load \(page 330\)](#)
- [Change processing \(page 330\)](#)
- [Data errors \(page 330\)](#)

Server utilization

The **Server Utilization** section contains the following graphs:

- **Average Memory** - The average memory consumed by each task.
- **Average Disk Utilization** - The average disk space utilized by each task.
- **Average Task CPU** - The average CPU (as a percentage of server CPU) consumed by each task.

Full load

The **Full Load** section contains the following graph:

- **Average Throughput** - The average target throughput (in rec/sec) for each task.

Change processing

The **Change Processing** section contains the following graphs:

- **Total Applied Changes** - The total number of applied changes for each task.
- **Average Target Throughput** - The average target throughput (in rec/sec) for each task.
- **Average Apply Latency**- The average apply latency (in seconds) for each task.
- **Total Change Volume** - The total COMMIT change volume for each task.

Data errors

The total number of data errors for each task, including both Full Load and Change Processing tasks.

Top servers

The Top Servers dashboard provides a summary of activity and performance metrics for the given time frame and filter set. It also provides a bubble chart comparing three key metrics for your top servers together with a corresponding data grid. You can customize the bubble chart by selecting the number of servers to view and the three metrics to plot, including the **Rank By** metric, the **X-Axis** metric and the **Y-Axis** metric. The **Rank By** selection in the bubble chart will also control the sorting of the data in the grid below. You can also choose whether to rank the bubble chart data in Ascending or Descending order. The grid can be further customized as described in [Customizing task columns \(page 249\)](#).

For the specified filters, the Top Servers dashboard displays the following key metrics:

- [Server utilization \(page 331\)](#)
- [Full load \(page 331\)](#)
- [Change processing \(page 331\)](#)

Server utilization

The **Server Utilization** section contains the following metrics:

- **Average Memory** - The average memory consumed by all tasks on a server.
- **Average Disk Utilization** - The average memory consumed by all tasks on a server.
- **Average Qlik CPU** - The average Qlik CPU (as a percentage) for each server.
- **Average Machine CPU** - The average machine CPU (as a percentage) for each server.
- **Average Task CPU** - The average CPU per task (as a percentage of server CPU) for each server.

Full load

The **Full Load** section contains the following metrics:

- **Total Tables:** The total number of tables for all tasks on a server.
- **Total Records:** The total number of records for all tasks on a server.
- **Max Load Duration:** The longest duration of the load process (in seconds).
- **Average Load Duration:** The average duration of the load process for all completed tasks on a server (in seconds).
- **Average Source Throughput (rec/sec):** The average source throughput (in rec/sec) per task.
- **Average Source Throughput (kb/sec):** The average source throughput (in kb/sec) per task.
- **Average Target Throughput (rec/sec):** The average target throughput (in rec/sec) per task.
- **Average Target Throughput (kb/sec):** The average target throughput (in kb/sec) per task.

Change processing

The **Change Processing** section contains the following metrics:

- **Total Applied Changes:** The total number of applied changes for all tasks on a server.
- **Total INSERTs:** The total number of INSERTs applied.
- **Total UPDATEs:** The total number of UPDATEs applied.
- **Total DELETEs:** The total number of DELETEs applied.

- **Total DDLs:** The total number of DDLs applied.
- **Total COMMIT Change Volume:** The total COMMIT change volume for all tasks on a server.
- **Total COMMIT Change Records:** The total number of COMMIT change records for all tasks on a server.
- **Total Applied Transactions:** The total number of COMMIT transactions for all tasks on a server.
- **Average Changes in Memory (Source):** The average number of changes accumulated in memory until source commit per task.
- **Average Changes on Disk (Source):** The average number of changes accumulated on disk until source commit per task.
- **Average Changes for Apply in Memory (Target):** The average number of changes in memory during apply and until target commit per task.
- **Average Changes for Apply on Disk (Target):** The average number of changes on disk during apply and until target commit per task.
- **Average Source Throughput (rec/sec):** The average source throughput (in rec/sec) per task.
- **Average Source Throughput (kb/sec):** The average source throughput (in kb/sec) per task.
- **Average Target Throughput (rec/sec):** The average target throughput (in rec/sec) per task.
- **Average Target Throughput (kb/sec):** The average target throughput (in kb/sec) per task.
- **Average Apply Latency (secs):** The average apply latency (in seconds) per task.
- **Average Source Latency (secs):** The average source latency (in seconds) per task.

Top tasks

The **Top Tasks** dashboard provides a summary of activity and performance metrics for the given time frame and filter set. It also provides a bubble chart comparing three key metrics for your top tasks together with a corresponding data grid. You can customize the bubble chart by selecting the number of tasks to view and the three metrics to plot, including the **Rank By** metric, the **X-Axis** metric and the **Y-Axis** metric. You can also choose whether to rank the bubble chart data in Ascending or Descending order. The grid can be further customized as described in [Customizing task columns \(page 249\)](#).

For the specified filters, the **Top Tasks** dashboard displays the following key metrics:

- [Server utilization \(page 332\)](#)
- [Full load \(page 332\)](#)
- [Change processing \(page 333\)](#)

Server utilization

The **Server Utilization** section contains the following metrics:

- **Average Memory** - The average memory consumed by a task.
- **Average Disk Utilization** - The average disk space utilization by a task.
- **Average Task CPU** - The average CPU (as a percentage of server CPU) consumed by each task.

Full load

The **Full Load** section contains the following metrics:

- **Total Tables:** The total number of tables for a task.
- **Total Records:** The total number of records for a task.
- **Max Load Duration:** The longest duration of the load process (in seconds).
- **Average Load Duration:** The average duration of the load process for a completed task (in seconds).
- **Average Source Throughput (rec/sec):** The average source throughput (in rec/sec) for a task.
- **Average Source Throughput (kb/sec):** The average source throughput (in kb/sec) for a task.
- **Average Target Throughput (rec/sec):** The average target throughput (in rec/sec) for a task.
- **Average Target Throughput (kb/sec):** The average target throughput (in kb/sec) for a task.

Change processing

The **Change Processing** section contains the following metrics:

- **Total Applied Changes:** The total number of applied changes for a task.
- **Total INSERTs:** The total number of INSERTs applied.
- **Total UPDATES:** The total number of UPDATES applied.
- **Total DELETES:** The total number of DELETES applied.
- **Total DDLs:** The total number of DDLs applied.
- **Total COMMIT Change Volume:** The total COMMIT change volume for a task.
- **Total COMMIT Change Records:** The total number of COMMIT change records for a task.
- **Total Applied Transactions:** The total number of COMMIT transactions for a task.
- **Average Changes in Memory (Source):** The average number of changes accumulated in memory until source commit for a task.
- **Average Changes on Disk (Source):** The average number of changes accumulated on disk until source commit for a task.
- **Average Changes for Apply in Memory (Target):** The average number of changes in memory during apply and until target commit for a task.
- **Average Changes for Apply on Disk (Target):** The average number of changes on disk during apply and until target commit for a task.
- **Average Source Throughput (rec/sec):** The average source throughput (in rec/sec) for a task.
- **Average Source Throughput (kb/sec):** The average source throughput (in kb/sec) for a task.
- **Average Target Throughput (rec/sec):** The average target throughput (in rec/sec) for a task.
- **Average Target Throughput (kb/sec):** The average target throughput (in kb/sec) for a task.
- **Average Apply Latency (secs):** The average apply latency (in seconds) for a task.
- **Average Source Latency (secs):** The average source latency (in seconds) for a task.

Capacity planning

The Capacity Planning dashboard is meant to be run for a small number of representative servers and Change Processing tasks to help you forecast sizing requirements. It provides an overview of key metrics related to activity and performance as well as providing a summary of key capacity indicator metrics.

Charts related to Server Utilization and Change Processing Performance are also shown for the given time frame. Below each chart, you can toggle to view the trending data by hour, by day, by week, by month, or by minute. For the specified filters, the Capacity Planning dashboard displays the following information and charts:

Overall activity

- **Time Frame Start and End:** The start and end time for the tasks that were run.
- **Total Tasks:** The total number of tasks that were run.
- **Total Tables:** The total number of tables for all tasks.
- **Total INSERTs:** The total number of INSERTs applied.
- **Total UPDATEs:** The total number of UPDATEs applied.
- **Total DELETEs:** The total number of DELETEs applied.
- **Total DDLs:** The total number of DDLs applied.
- **Total Applied Changes:** The total number of applied changes for all tasks.
- **Total COMMIT Change Volume:** The total COMMIT change volume for all tasks.
- **Total COMMIT Change Records:** The total number of COMMIT change records for all tasks.
- **Total Applied Transactions:** The total number of COMMIT transactions for all tasks.
- **Applied Changes by Task:** Pie chart showing the total number of applied changes by task.
- **Applied Transactions by Task:** Pie chart showing the total number of COMMIT transactions by task.
- **COMMIT Change Records by Task:** Pie chart showing the total number of COMMIT change records by task.
- **COMMIT Change Volume by Task:** Pie chart showing the total COMMIT change volume by task.

Overall performance

- **Average Memory:** The average memory per task.
- **Average Disk Usage:** The average disk space utilization per task.
- **Avg Qlik CPU -** The average Qlik CPU (as a percentage) per server.
- **Avg Machine CPU -** The average machine CPU (as a percentage) per server.
- **Throughput:** The average, minimum and maximum target throughput (in rec/sec) per task.
- **Latency:** The average, minimum and maximum apply latency (in seconds) per task.

Capacity indicators

- **Average Tables per Task:** The average number of tables per task.
- **Average Applied Changes per Task:** The average number of applied changes per task.
- **Average Applied Transactions per Task:** The average number of COMMIT transactions per task.
- **Average Transaction Size:** The average COMMIT change volume per COMMIT transaction.
- **Average Number of Changes per Transaction:** The average number of applied changes per COMMIT transaction.

Server utilization

- **Average Memory:** The area chart shows the average memory consumed by all tasks. The lines show the average memory for each task that was run.
- **Average Task CPU:** The area chart shows the average task CPU (as a percentage). The lines show the average CPU for each task that was run.
- **Total Applied Changes and Change Volume:** The total number of applied changes as well as the total COMMIT change volume for all tasks.
- **Average Disk Usage:** The area chart shows the average disk space utilization for all tasks. The lines show the average disk usage for each task that was run.
- **Changes in Memory and Disk:** The average number of changes accumulated in memory and on disk until source commit as well as the average number of changes in memory and on disk during apply and until target commit per task.

Change processing performance

- **Applied Changes:** The total number of applied changes for all tasks.
- **Throughput and Latency:** The average source and target throughput (in rec/sec) and the average source and apply latency (in seconds) per task.

15.4 Exporting to TSV

For each dashboard, you can export the data to a TSV file.

To export Trends, Trends by Server, Trends by Task and Capacity Planning data:

1. Click the **Export to TSV** button to the right of the **Filters** button.
The **Export to TSV** window opens.
2. Optionally, move the slider to change the default resolution level.
3. Select which charts to export. A separate file will be created for each of the selected charts.
4. Click **Export**.
Depending on your browser settings, the file will either be exported (downloaded) to your browser's default "Downloads" location or you will be prompted to save the file to your preferred location.

To export Top Servers and Top Tasks grid data:

- Click the **Export to TSV** button to the right of the **Filters** button.
Depending on your browser settings, the file will either be exported (downloaded) to your browser's default "Downloads" location or you will be prompted to save the file to your preferred location.

15.5 Creating filters

You can filter the dashboards to only show data for a particular timeframe and/or for particular objects (tasks, Replicate servers, source endpoints, and target endpoints).

To create a timeframe filter:

1. Click the  icon to the right of the **From** field and select a "from" date using the calendar control.
OR
Enter the date manually.
2. Specify a "from" time in the two fields to the right of the  icon.
3. Click the  icon to the right of the **To** field and select a "to" date using the calendar control.
OR
Enter the date manually.
4. Specify a "to" time in the two fields to the right of the  icon.
5. Click the **Apply** toolbar button.

To create an object filter

1. Click the **Filters** toolbar button.
The **Filters** window opens.
2. From the left pane, choose one of the following object types:
 - Servers
 - Source endpoints
 - Target endpoints
 - Tasks



By default, all instances of the selected object type are shown.

3. To search for a specific object, in the **Name** field, enter all or part of the object name and then click **Search**.
4. If you selected **Source endpoints** or **Target endpoints**, optionally filter the search results by **Type** as well.
If you do not select an endpoint type, all endpoints will be shown in the **Source Endpoint List/Target Endpoint List**.
5. If you selected **Tasks**, optionally filter the search results by **Task Type** and/or **Status** as well.
If you do not select a type, all tasks will be shown in the **Task List**.
6. To exclude a specific object, select the object(s) and click the **Exclude** button (multiple selection is supported). The object is added to the list on the right.
7. To include a specific object, select the object(s) and click the **Include** button (multiple selection is supported). The object is added to the list on the right.
8. To include or exclude all objects that match a pattern, type the pattern in the **Name** field and then click the **Include** or **Exclude** button accordingly.

For example, to include all task names that start with `bus`, type `bus*`.
The pattern is added to the **<Object_Name> Selection Patterns** list on the right.



The patterns list can contain both include and exclude patterns.

- Click **OK** to save your settings and then click **Apply** to apply the filter.

15.6 Using the Pivot Menu

The Pivot Menu is located on the right of the **Analytics** tab. You can select any item in a chart or any row from a grid, and the pivot menu will open up allowing you to pivot, or drill, to another dashboard. When you pivot/drill to another dashboard, the selected item(s) will be applied as filters to the selected dashboard.

15.7 Analytics repository schema

In addition to the data provided by the Enterprise Manager Analytics dashboards, organizations can also use their own BI tools to generate reports based on the available metrics. To assist with this, the following topic provides a description of the Analytics repository schema together with a [couple of sample queries](#).

The Analytics repository contains the following tables:

- [aem_endpoint_type \(page 337\)](#)
- [aem_meta_source_database \(page 338\)](#)
- [aem_meta_target_database \(page 338\)](#)
- [aem_server](#)
- [aem_source_database](#)
- [aem_target_database](#)
- [aem_target_processes](#)
- [aem_task](#)
- [aem_task_name](#)
- [aem_task_previous_metrics](#)
- [aem_task_profile](#)
- [aem_task_state](#)
- [aem_task_stop_reason](#)

aem_endpoint_type

The `aem_endpoint_type` table provides lookup information about the Replicate endpoints.

Available table columns

Column	Data Type	Description
<code>endpoint_type_id</code>	Integer	The endpoint type ID.
<code>endpoint_type_name</code>	Character varying(1024)	The endpoint type.

aem_meta_source_database

The aem_meta_source_database table provides lookup information about the source endpoint.

Available table columns

Column	Data Type	Description
meta_source_database_id	Integer	The source endpoint ID.
meta_source_database_name	Character varying(1024)	The source endpoint name.
endpoint_type_id	Integer	The endpoint type ID.

aem_meta_target_database

The aem_meta_target_database table provides lookup information about the target endpoint.

Available table columns

Column	Data Type	Description
meta_target_database_id	Integer	Target Endpoint ID
meta_target_database_name	Character varying(1024)	Target Endpoint Name
endpoint_type_id	Integer	Endpoint Type ID

aem_server

The aem_server table provides lookup information about the Replicate server.

Available table columns

Column	Data Type	Description
server_id	Integer	The Replicate server ID.
server_name	Character varying(1024)	The Replicate server name.
Host	Character varying(1024)	The host name or IP address.
port	Integer	The Replicate port.
platform	Character varying(1024)	The Replicate server platform.
version	Character varying(1024)	The Replicate version.

aem_source_database

The aem_source_database table provides information about the source endpoint used in a specific task.

Available table columns

Column	Data Type	Description
task_id	Integer	The task run ID.

Column	Data Type	Description
source_database_id	Integer	The source endpoint ID.
source_database_name	Character varying(1024)	The source endpoint name.
endpoint_type_id	Integer	The endpoint Type ID.

aem_target_database

The aem_target_database table provides information about the target endpoint used in a specific task.

Available table columns

Column	Data Type	Description
task_id	Integer	The task run ID.
target_database_id	Integer	The target endpoint ID.
target_database_name	Character varying(1024)	The target endpoint name.
endpoint_type_id	Integer	The target endpoint type ID.

aem_target_processes

The aem_target_processes table is used internally to monitor the Enterprise Manager Analytics processes such as the Collector and the Purger.

aem_task

This is a fact table for each run or instance of a Replicate task.

Available table columns

Column	Data Type	Description
task_id	Integer	The task run ID.
target_database_id	Integer	The target endpoint ID.
target_database_name	Character varying (1024)	Target endpoint name.
endpoint_type_id	Integer	The endpoint type ID.
retrieval_time	Timestamp without time zone	The time that the information about the task was retrieved.
server_id	Integer	The Replicate server ID.
task_name_id	Integer	The task name ID.
task_state_id	Integer	The task state ID.
task_stop_reason_id	Integer	The task stop Reason ID.

Column	Data Type	Description
task_profile_id	Integer	The task profile ID.
cdc_evt_applied_insert_count	Integer	The number of INSERTs since the last retrieval time or since the task was started.
cdc_evt_applied_update_count	Integer	The number of UPDATES since the last retrieval time or since the task was started.
cdc_evt_applied_delete_count	Integer	The number of DELETES since the last retrieval time or since the task was started.
cdc_evt_applied_ddl_count	Integer	The number of DDLs since the last retrieval time or since the task was started.
full_load_tables_completed_count	Integer	The number of completed tables loaded to the Target at retrieval time.
full_load_tables_loading_count	Integer	The number of tables being loaded to the target at retrieval time.
full_load_tables_queued_count	Integer	The number of tables waiting to be loaded to the target at retrieval time.
full_load_tables_with_error_count	Integer	The number of tables that could not be loaded to the target at retrieval time due to an error.
full_load_total_records_transferred	Integer	The total number of records that have completed loading to the target at retrieval time.
full_load_est_records_count_for_all_tables	Integer	The estimated number of records to be loaded to the target.
full_load_completed	Integer	Indicates if the Full Load has completed.
full_load_start	Timestamp without time zone	The start time of the Full Load.
full_load_finish	Timestamp without time zone	The finish time of the Full Load.
full_load_thrput_src_thrput_records_count	Integer	The Full Load source throughput (in rec/sec) at retrieval time.
full_load_thrput_src_thrput_volume	Integer	The Full Load source throughput (in kb/sec) at retrieval time.
full_load_thrput_trg_thrput_records_count	Integer	The Full Load target throughput (in rec/sec) at retrieval time.
full_load_thrput_trg_thrput_volume	Integer	The Full Load target throughput (in kb/sec) at retrieval time.

Column	Data Type	Description
cdc_thrput_src_thrput_records_count	Integer	The Change Processing source throughput (in rec/sec) at retrieval time.
cdc_thrput_src_thrput_volume	Integer	The Change Processing source throughput (in kb/sec) at retrieval time.
cdc_thrput_trg_thrput_records_count	Integer	The Change Processing target throughput (in rec/sec) at retrieval time.
cdc_thrput_trg_thrput_volume	Integer	The Change Processing target throughput (in kb/sec) at retrieval time
cdc_trans_read_rollback_count	Integer	The number of ROLLBACK transactions since the last retrieval time or since the task was started.
cdc_trans_read_records_rollback_count	Integer	The number of ROLLBACK change records since the last retrieval time or since the task was started.
cdc_trans_rollback_change_volume	Integer	The volume of ROLLBACK changes (in bytes).
cdc_trans_applied_transactions_in_progress_count	Integer	The number of transactions in progress at retrieval time.
cdc_trans_applied_records_in_progress_count	Integer	The number of records for all transactions in progress at retrieval time.
cdc_trans_applied_comitted_transaction_count	Integer	The number of transactions committed since the last retrieval time or since the task was started.
cdc_trans_applied_records_comitted_count	Integer	The number of records for all committed transactions since the last retrieval time or since the task was started.
cdc_trans_applied_volume_comitted	Integer	The volume of change for all committed transactions (in bytes) since the last retrieval time or since the task was started.
cdc_trans_read_memory_events_count	Integer	The number of changes accumulated in memory until source commit at retrieval time.
cdc_trans_read_swapped_events_count	Integer	The number of changes accumulated on disk until source commit at retrieval time.
cdc_trans_applied_memory_events_count	Integer	The number of changes in memory during apply and until target commit at retrieval time

Column	Data Type	Description
cdc_trans_applied_swap_events_count	Integer	The number of changes on disk during apply and until target commit at retrieval time.
cdc_source_latency	Integer	The average time gap between the original change in the source endpoint and capturing it (in seconds) at retrieval time.
cdc_apply_latency	Integer	The overall latency (in seconds) at retrieval time.
memory_usage_kb	Integer	The memory usage for the task (in kilobytes) at retrieval time.
disk_usage_kb	Integer	The utilization of disk space for the task (in kilobytes) at retrieval time.
cpu_percentage	Integer	The CPU consumption of the task (as a percentage of server CPU with a value of 0-100) at retrieval time.
data_error_count	Integer	The total number of data errors at retrieval time for all tables involved in a task.
task_option_full_load_enabled	Integer	Indicates if Full Load is enabled.
task_option_apply_changes_enabled	Integer	Indicates if Apply Changes is enabled.
task_option_store_changes_enabled	Integer	Indicates if Store Changes is enabled.
task_option_audit_changes_enabled	Integer	Indicates if Audit Changes is enabled.
task_option_recovery_enabled	Integer	Indicates if Recovery is enabled.
server_cpu_percentage	Integer	The CPU percentage being utilized by the Replicate server process running on the server machine at retrieval time.
machine_cpu_percentage	Integer	The total CPU percentage being utilized by all processes (i.e. not just Qlik processes) running on the server machine at retrieval time.
tasks_cpu_percentage	Integer	The total CPU percentage being utilized by all Replicate task processes running on the server machine at retrieval time.

aem_task_name

The aem_task_name table provides lookup information about the task name.

Available table columns

Column	Data Type	Description
server_id	Integer	The Replicate server ID.
task_name_id	Integer	The task name ID.
task_name	Character varying(1024)	The task name.

aem_task_previous_metrics

Used internally to calculate differences between the current run of a task and the previous run.

aem_task_profile

The aem_task_profile table provides lookup information about the task profile.

Available table columns

Column	Data Type	Description
task_profile_id	Integer	The task profile ID.
name	Character varying(1024)	The task profile name.
description	Character varying(1024)	The task profile description.

aem_task_state

The aem_task_state table provides lookup information about the task state.

Available table columns

Column	Data Type	Description
task_state_id	Integer	The task state ID.
name	Character varying(1024)	The task state name.
description	Character varying(1024)	The task state description.

aem_task_stop_reason

The aem_task_stop_reason table provides lookup information about the reason that a task stopped.

Available table columns

Column	Data Type	Description
task_stop_reason_id	Integer	The stop reason ID.
name	Character varying(1024)	The stop reason name.
description	Character varying(1024)	The stop reason description.

Sample Queries

The following sample queries demonstrate how useful information can be extracted from the schema.

Query 1:

Provides a sampling of Full Load and Change Processing metrics for all tasks with a source endpoint named "Teradata DB" for the month of August 2017.

```

SELECT
  AEM_SERVER.SERVER_NAME AS "Server"
  ,AEM_TASK_NAME.TASK_NAME AS "Task"
  ,AVG(AEM_TASK.MEMORY_USAGE_KB / 1024::float) AS "Avg Memory"
  ,AVG(AEM_TASK.DISK_USAGE_KB) / 1024::float AS "Avg IO"
  ,MAX((CASE WHEN AEM_TASK.FULL_LOAD_TABLES_COMPLETED_COUNT IS NULL THEN 0
ELSE AEM_TASK.FULL_LOAD_TABLES_COMPLETED_COUNT END)+(CASE WHEN AEM_TASK.FULL_
LOAD_TABLES_LOADING_COUNT IS NULL THEN 0 ELSE AEM_TASK.FULL_LOAD_TABLES_
LOADING_COUNT END)+(CASE WHEN AEM_TASK.FULL_LOAD_TABLES_QUEUED_COUNT IS NULL
THEN 0 ELSE AEM_TASK.FULL_LOAD_TABLES_QUEUED_COUNT END)+(CASE WHEN AEM_
TASK.FULL_LOAD_TABLES_WITH_ERROR_COUNT IS NULL THEN 0 ELSE AEM_TASK.FULL_
LOAD_TABLES_WITH_ERROR_COUNT END)) AS "Full Load Total Tables"
  ,MAX(FULL_LOAD_TOTAL_RECORDS_TRANSFERRED) AS "Full Load Total Records"
  ,MAX(AEM_TASK.FULL_LOAD_FINISH-AEM_TASK.FULL_LOAD_START) AS "Full Load Max
Load Duration"
  ,AVG(AEM_TASK.FULL_LOAD_FINISH-AEM_TASK.FULL_LOAD_START) AS "Full Load Avg
Load Duration"
  ,AVG((CASE WHEN AEM_TASK.FULL_LOAD_THRPUT_TRG_THRPUT_RECORDS_COUNT IS NULL
THEN 0 ELSE AEM_TASK.FULL_LOAD_THRPUT_TRG_THRPUT_RECORDS_COUNT END)) AS "Full
Load Avg Target Throughput Records per Second"
  ,SUM((CASE WHEN AEM_TASK.CDC_EVT_APPLIED_INSERT_COUNT IS NULL THEN 0 ELSE
AEM_TASK.CDC_EVT_APPLIED_INSERT_COUNT END)+(CASE WHEN AEM_TASK.CDC_EVT_
APPLIED_UPDATE_COUNT IS NULL THEN 0 ELSE AEM_TASK.CDC_EVT_APPLIED_UPDATE_
COUNT END)+(CASE WHEN AEM_TASK.CDC_EVT_APPLIED_DELETE_COUNT IS NULL THEN 0
ELSE AEM_TASK.CDC_EVT_APPLIED_DELETE_COUNT END)+(CASE WHEN AEM_TASK.CDC_EVT_
APPLIED_DDL_COUNT IS NULL THEN 0 ELSE AEM_TASK.CDC_EVT_APPLIED_DDL_COUNT
END)) AS "CDC Total Applied Changes"
  ,SUM((CASE WHEN AEM_TASK.CDC_TRANS_APPLIED_COMMITTED_TRANSACTION_COUNT IS
NULL THEN 0 ELSE AEM_TASK.CDC_TRANS_APPLIED_COMMITTED_TRANSACTION_COUNT END))
AS "CDC Total Applied Transactions"
  ,AVG((CASE WHEN AEM_TASK.CDC_THRPUT_TRG_THRPUT_RECORDS_COUNT IS NULL THEN 0
ELSE AEM_TASK.CDC_THRPUT_TRG_THRPUT_RECORDS_COUNT END)) AS "CDC Avg Target
Throughput Records per Second"
  ,AVG((CASE WHEN AEM_TASK.CDC_APPLY_LATENCY IS NULL THEN 0 ELSE AEM_
TASK.CDC_APPLY_LATENCY END)) AS "CDC Avg Apply Latency"
FROM
  AEM_TASK
  INNER JOIN AEM_SERVER ON (AEM_TASK.SERVER_ID = AEM_SERVER.SERVER_ID)
  INNER JOIN AEM_TASK_NAME ON (AEM_TASK.TASK_NAME_ID = AEM_TASK_NAME.TASK_
NAME_ID)
  INNER JOIN AEM_SOURCE_DATABASE ON (AEM_TASK.ID = AEM_SOURCE_DATABASE.TASK_
ID)

```

```

WHERE
    (AEM_TASK.RETRIEVAL_TIME >= TIMESTAMP WITH TIME ZONE '2017-08-01
00:00:00.00000+00')
    AND (AEM_TASK.RETRIEVAL_TIME < TIMESTAMP WITH TIME ZONE '2017-09-01
00:00:00.00000+00')
    AND (AEM_SOURCE_DATABASE.SOURCE_DATABASE_NAME = 'Teradata DB')
GROUP BY
    AEM_SERVER.SERVER_NAME
    ,AEM_TASK_NAME.TASK_NAME

```

Query 2:

Provides a trend throughout the day of the number of Completed, Queued, Loading and Error tables related to Full Load for the month of August 2017 and for a Replicate server named "rep-server1-prod".

```

SELECT
    SUM((CASE WHEN AEM_TASK.FULL_LOAD_TABLES_COMPLETED_COUNT IS NULL THEN 0
ELSE AEM_TASK.FULL_LOAD_TABLES_COMPLETED_COUNT END)) AS "Full Load Tables
Completed"
    ,SUM((CASE WHEN AEM_TASK.FULL_LOAD_TABLES_LOADING_COUNT IS NULL THEN 0 ELSE
AEM_TASK.FULL_LOAD_TABLES_LOADING_COUNT END)) AS "Full Load Tables Loading"
    ,SUM((CASE WHEN AEM_TASK.FULL_LOAD_TABLES_QUEUED_COUNT IS NULL THEN 0 ELSE
AEM_TASK.FULL_LOAD_TABLES_QUEUED_COUNT END)) AS "Full Load Tables Queued"
    ,SUM((CASE WHEN AEM_TASK.FULL_LOAD_TABLES_WITH_ERROR_COUNT IS NULL THEN 0
ELSE AEM_TASK.FULL_LOAD_TABLES_WITH_ERROR_COUNT END)) AS "Full Load Tables
Error"
    ,AEM_TASK.RETRIEVAL_TIME AS "Date and Time"
FROM
    AEM_TASK
    INNER JOIN AEM_SERVER ON (AEM_TASK.SERVER_ID = AEM_SERVER.SERVER_ID)
WHERE
    (AEM_TASK.RETRIEVAL_TIME >= TIMESTAMP WITH TIME ZONE '2017-08-01
00:00:00.00000+00')
    AND (AEM_TASK.RETRIEVAL_TIME < TIMESTAMP WITH TIME ZONE '2017-09-01
00:00:00.00000+00')
    AND (AEM_SERVER.SERVER_NAME IN ('rep-server1-prod'))
GROUP BY
    AEM_TASK.RETRIEVAL_TIME
ORDER BY
    "Date and Time" ASC

```

Query 3:

Provides an daily trend of Average Throughput and Latency metrics related to Change Processing for the month of August 2017 and for a Replicate server named "rep-server1-prod".

```

SELECT
  AVG((CASE WHEN AEM_TASK.CDC_THRPUT_SRC_THRPUT_RECORDS_COUNT IS NULL THEN 0
  ELSE AEM_TASK.CDC_THRPUT_SRC_THRPUT_RECORDS_COUNT END)) AS "CDC Avg Source
  Throughput Records"
  ,AVG((CASE WHEN AEM_TASK.CDC_THRPUT_TRG_THRPUT_RECORDS_COUNT IS NULL THEN 0
  ELSE AEM_TASK.CDC_THRPUT_TRG_THRPUT_RECORDS_COUNT END)) AS "CDC Avg Target
  Throughput Records"
  ,AVG((CASE WHEN AEM_TASK.CDC_SOURCE_LATENCY IS NULL THEN 0 ELSE AEM_
  TASK.CDC_SOURCE_LATENCY END)) AS "CDC Avg Source Latency"
  ,AVG((CASE WHEN AEM_TASK.CDC_APPLY_LATENCY IS NULL THEN 0 ELSE AEM_
  TASK.CDC_APPLY_LATENCY END)) AS "CDC Avg Apply Latency"
  ,EXTRACT(YEAR from AEM_TASK.RETRIEVAL_TIME) AS "Year"
  ,EXTRACT(MONTH from AEM_TASK.RETRIEVAL_TIME) AS "Month"
  ,EXTRACT(DAY from AEM_TASK.RETRIEVAL_TIME) AS "Day"
FROM
  AEM_TASK
  INNER JOIN AEM_SERVER ON (AEM_TASK.SERVER_ID = AEM_SERVER.SERVER_ID)
WHERE
  (AEM_TASK.RETRIEVAL_TIME >= TIMESTAMP WITH TIME ZONE '2017-08-01
  00:00:00.00000+00')
  AND (AEM_TASK.RETRIEVAL_TIME < TIMESTAMP WITH TIME ZONE '2017-09-01
  00:00:00.00000+00')
  AND (AEM_SERVER.SERVER_NAME IN ('rep-server1-prod'))
GROUP BY
  EXTRACT(YEAR from AEM_TASK.RETRIEVAL_TIME)
  ,EXTRACT(MONTH from AEM_TASK.RETRIEVAL_TIME)
  ,EXTRACT(DAY from AEM_TASK.RETRIEVAL_TIME)
ORDER BY
  "Year", "Month", "Day"

```

A Setting up High Availability

This section describes how to set up Enterprise Manager in a Windows High Availability Cluster.



- When the Enterprise Manager data directory is installed in a non-default location, the `-d <data_directory>` parameter must be included in all commands, where `data_directory` is the location of the **data** directory.
- To obtain a Replication Management license for Enterprise Manager in a High Availability Cluster, you must provide your Qlik Sales Representative with the official cluster FQDN.
- Throughout this chapter "Node 1" and "Node 2" are used to refer to the primary and secondary clustering nodes respectively.



When building failover cluster solutions with Replicate using Windows Server Failover Cluster (WSFC) or a Linux failover cluster software, Qlik recommends using a block device (physical, virtual or iSCSI-based) for the shared Replicate DATA folder. Using NFS or SMB-based storage is not supported due to the associated latency which could greatly degrade the data transfer performance, as well as due to reduced reliability and compatibility issues. When building a cloud-based high availability solution that needs to span different availability zones, it is recommended to use a Storage-as-a-Service solution that can handle the block-level replication of the storage and that is integrated with the chosen failover clustering software.

In this section:

- [Installing Qlik Enterprise Manager in a Windows cluster \(page 347\)](#)
- [Upgrading Qlik Enterprise Manager in a Windows cluster \(page 350\)](#)
- [Uninstalling Qlik Enterprise Manager from a Windows cluster \(page 352\)](#)

A.1 Installing Qlik Enterprise Manager in a Windows cluster

This topic explains how to install Enterprise Manager in a Windows cluster.

- For information on obtaining a license for Enterprise Manager in a Windows cluster, see [Replication Management license \(page 17\)](#).
- For information on upgrading and uninstalling Enterprise Manager in/from a Windows cluster, see [Upgrading Qlik Enterprise Manager in a Windows cluster \(page 350\)](#) and [Uninstalling Qlik Enterprise Manager from a Windows cluster \(page 352\)](#) respectively.
- For more information on the Master User Key commands mentioned in this section, see [Replacing the Master User Password \(page 45\)](#) and [Configuring Enterprise Manager using the CLI \(page 313\)](#).



Unless specified otherwise, all commands should be run from the Enterprise Manager bin folder. The default location is:

C:\Program Files\Attunity\Enterprise Manager\bin

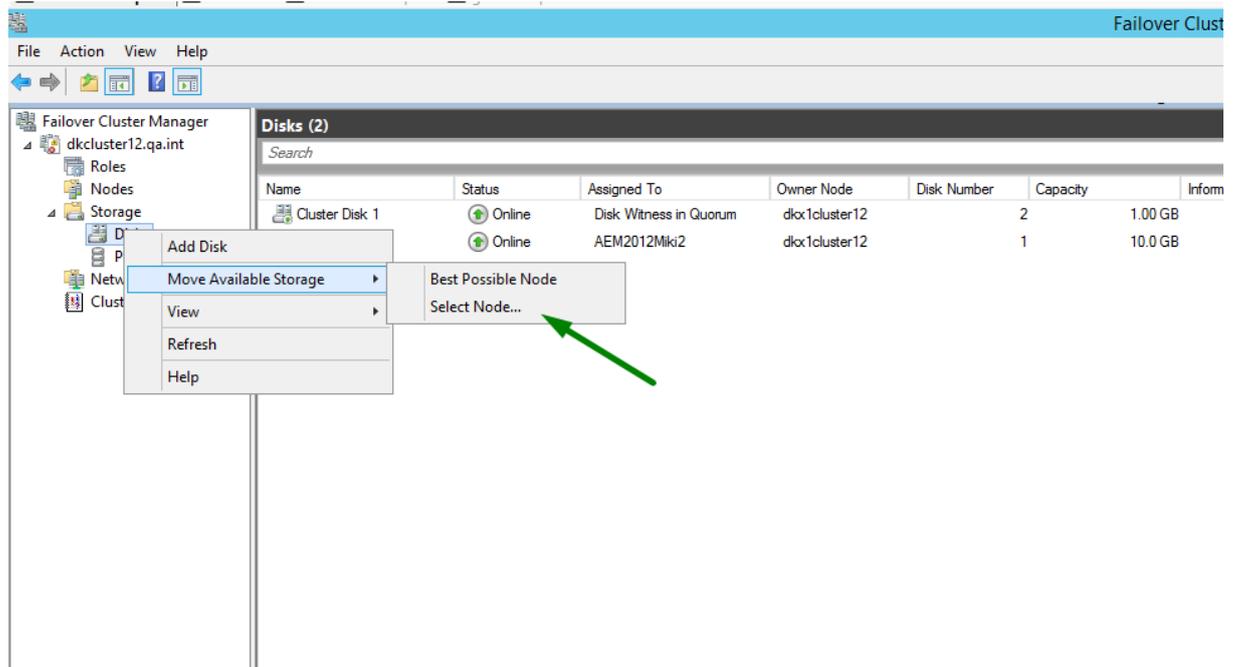
To install Enterprise Manager in a Windows cluster:

1. Open Failover Cluster Manager and connect to a network that contains the cluster nodes and a clustered disk (for the Enterprise Manager "data" folder).
2. Install Enterprise Manager on Node 1. This node should be the Cluster Disk owner. You can specify any **Destination Location**, but the "data" folder must be installed on a clustered disk (i.e. a disk that is accessible to the other cluster node).

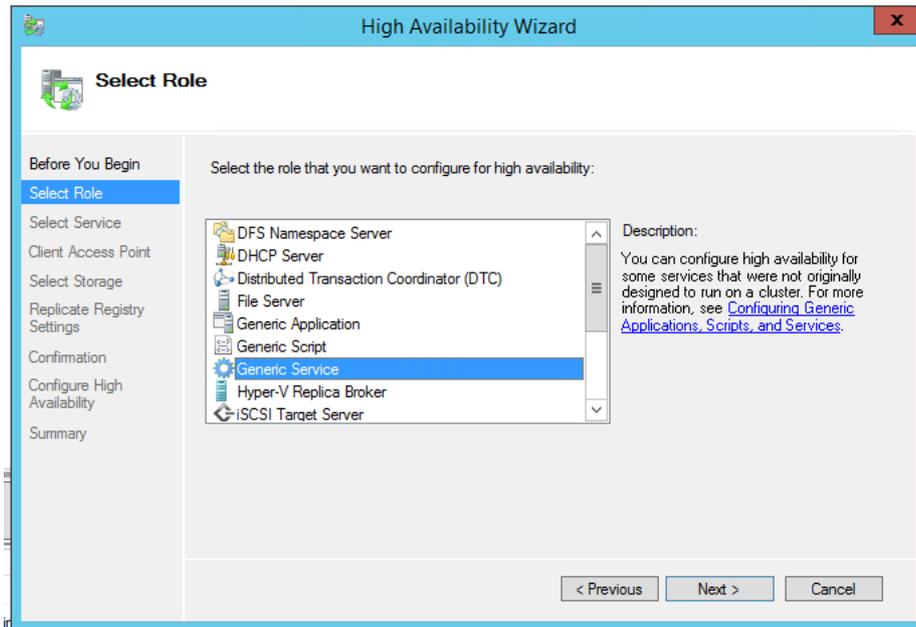


The "data" folder contains a "java" subfolder. At various points in the cluster setup, you will need to specify the location of both the "data" folder and the "java" folder. It is therefore recommended to make a note of the "data" folder location.

3. Set the Enterprise Manager Master User Key by opening a command prompt and running the following command:
`aemctl.exe -d data_folder_path masterkey set -p password`
where -d data_folder_path is the path to the shared Enterprise Manager **data** folder.
4. Set the Java Master User Key by opening a command prompt and running the following command from the Java **bin** folder (<INSTALL_DIR>\Enterprise Manager\java\bin by default):
`atajs.bat -d java_data_folder_path masterkey set password`
where -d java_data_folder_path is the path to the shared Enterprise Manager **data\java** folder.
5. On Node 1, open the **Windows Services** console and stop the Enterprise Manager service.
6. Move the shared Cluster Disk to Node 2.



7. Install Enterprise Manager on Node 2. The **Destination Location** can be any local folder, but make sure to install the "data" folder on the same clustered disk specified in [step 2 above](#).
8. Make sure the Enterprise Manager service is stopped on Node 2.
9. Set the Enterprise Manager Master User Key by opening a command prompt and running the following command:
`aemctl.exe -d data_folder_path masterkey set -p password`
where `-d data_folder_path` is the path to the shared Enterprise Manager **data** folder.
10. Set the Java Master User Key by opening a command prompt and running the following command from the Java **bin** folder (`<INSTALL_DIR>\Enterprise Manager\java\bin` by default):
`atajs.bat -d java_data_folder_path masterkey set password`
where `-d java_data_folder_path` is the path to the shared Enterprise Manager **data\java** folder.
11. Configure the Enterprise Manager cluster role as follows:
 - a. Right click on **Roles** and select **Configure Role**.
 - b. In the **Select Role:** Choose "Generic Service" and then click **Next**.



- c. In the **Select Service** screen, choose "Enterprise Manager". Click **Next**.
 - d. In the **Client Access Point** screen, enter the Enterprise Manager host name without the domain information (e.g. mycompany and not mycompany.qa.int). Click **Next**.
 - e. In the **Select Storage**, select the location on the clustered disk where the Enterprise Manager "data" folder was installed. Click **Next**.
 - f. Continue clicking **Next** until the **Finish** button is displayed and then click **Finish**.
12. Configure the host name by opening a command prompt and running the following command:
aemctl.exe -d data_folder_path configuration set -a host_name
where -d data_folder_path is the path to the shared Enterprise Manager **data** folder.



The host name must be specified as a FQDN (e.g. mycompany.qa.int and not mycompany)

13. **Enterprise Manager Cluster Role Dependencies:** Verify that the Enterprise Manager cluster role has set dependency on the clustered disk and the Enterprise Manager service. You can view the dependencies in the role properties.
Your Enterprise Manager cluster should now be ready for use.

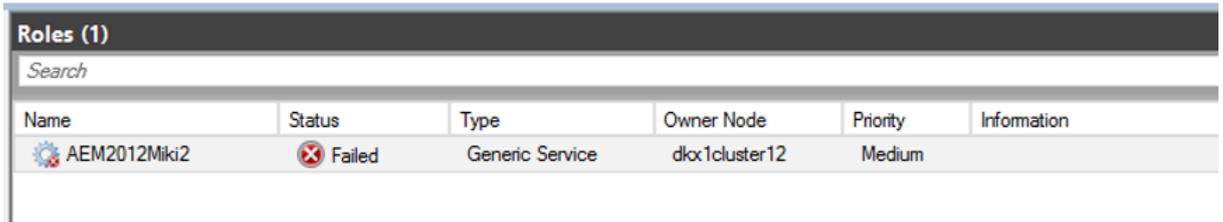
A.2 Upgrading Qlik Enterprise Manager in a Windows cluster

This topic explains how to upgrade Enterprise Manager in a Windows cluster.

To upgrade Enterprise Manager in a Windows cluster:

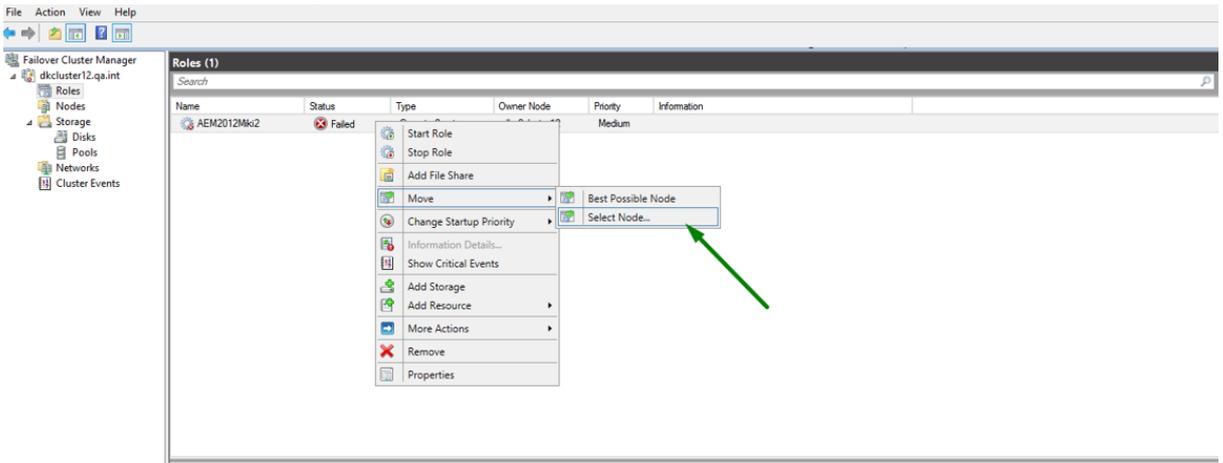
1. Open Failover Cluster Manager on the active cluster node.
2. Stop the Enterprise Manager service on both nodes.

- Verify that the Enterprise Manager Cluster Role status is "Failed".

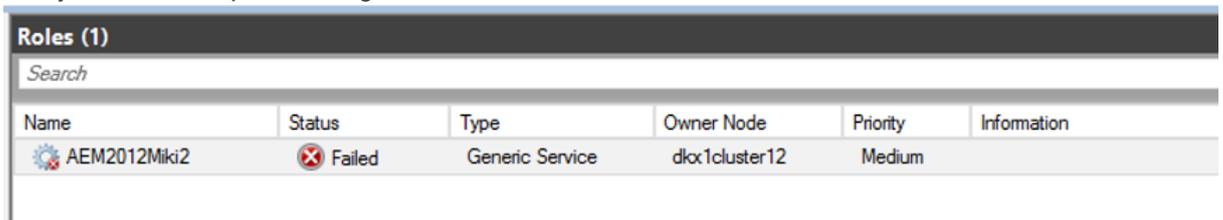


Name	Status	Type	Owner Node	Priority	Information
AEM2012Mki2	Failed	Generic Service	dkx1cluster12	Medium	

- Run the upgrade on Node 1.
- Move the Enterprise Manager Role to Node 2.

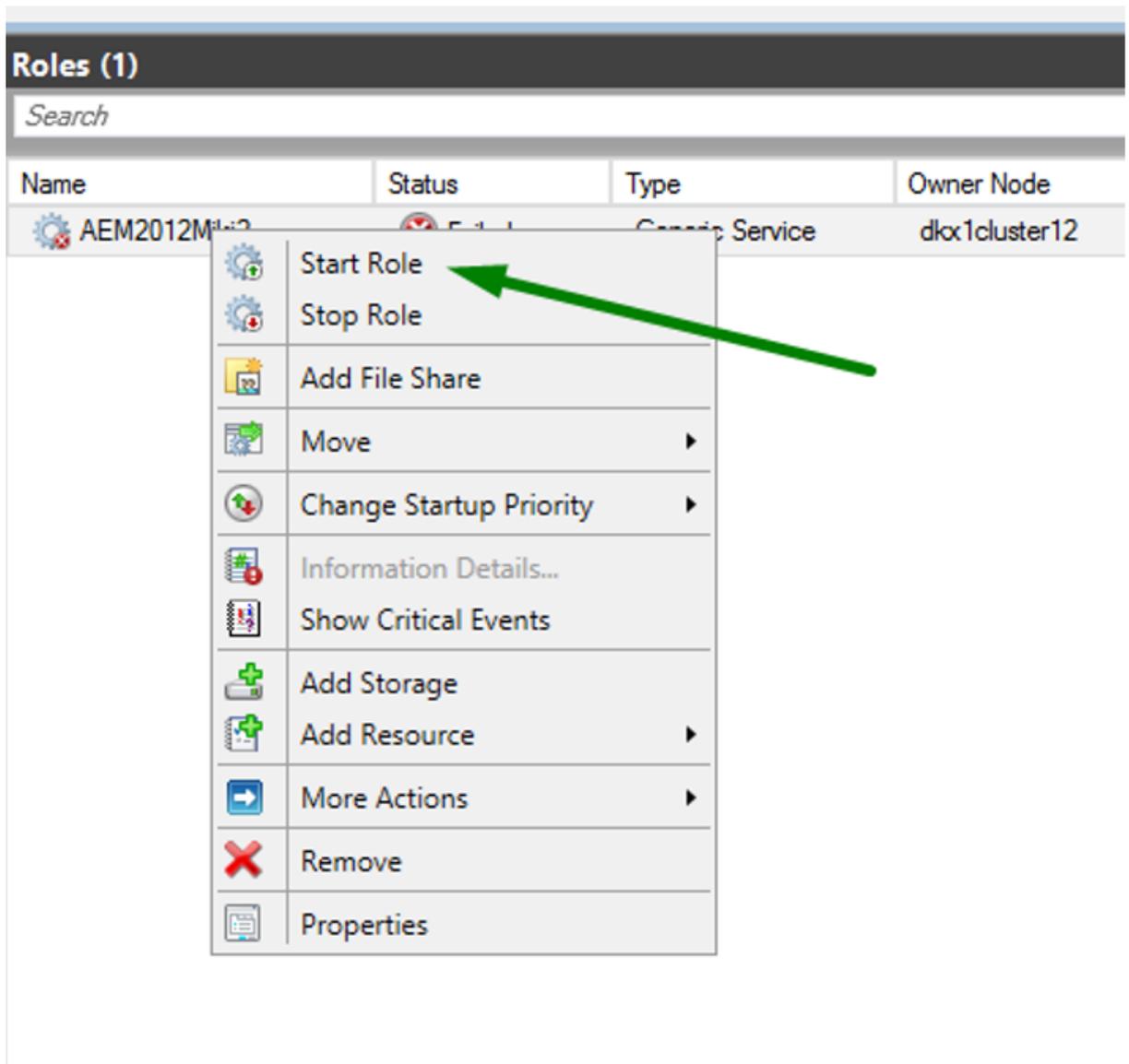


- Stop the Enterprise Manager service on both nodes.
- Verify that the Enterprise Manager Cluster Role status is "Failed".



Name	Status	Type	Owner Node	Priority	Information
AEM2012Mki2	Failed	Generic Service	dkx1cluster12	Medium	

- Run the upgrade on Node 2.
- Start the Enterprise Manager Cluster Role.



The upgrade should now be complete.

A.3 Uninstalling Qlik Enterprise Manager from a Windows cluster

This topic explains how to uninstall Enterprise Manager from a Windows cluster.

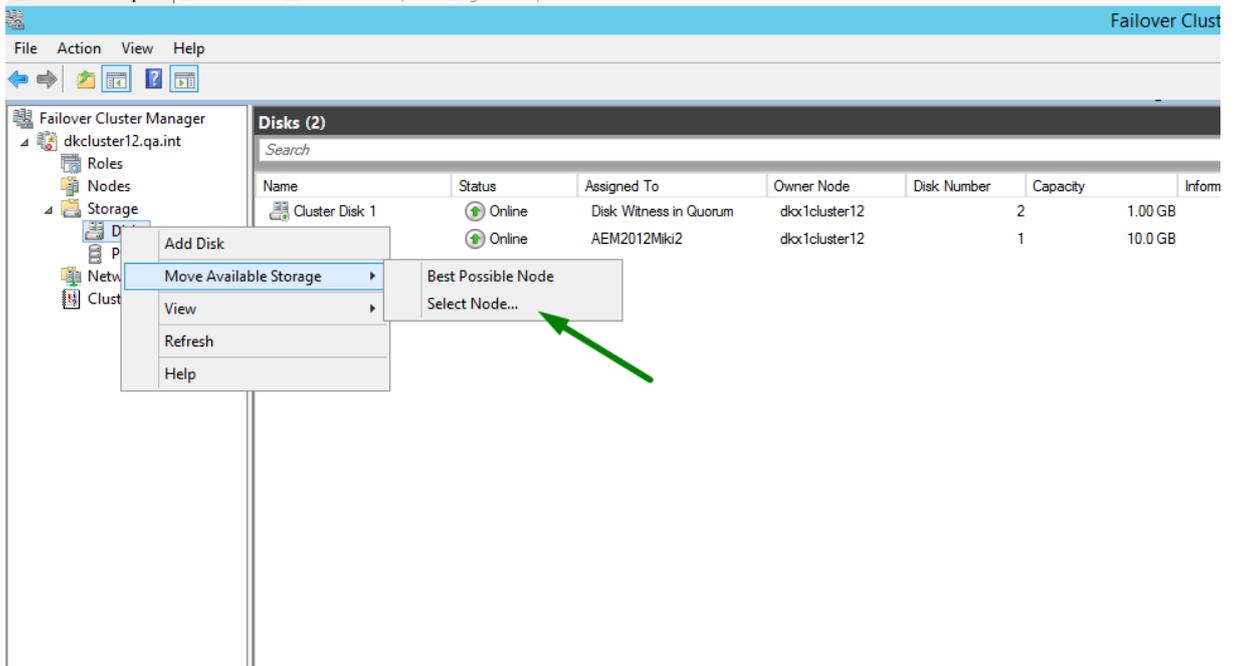
To uninstall Enterprise Manager from a Windows cluster:

1. Open Failover Cluster Manager on the active cluster node.
2. On each of the cluster nodes, open the **Windows Services** console and stop the Enterprise Manager service.
3. Verify in Failover Cluster Manager that the status of the Enterprise Manager cluster role is "Failed".

A Setting up High Availability

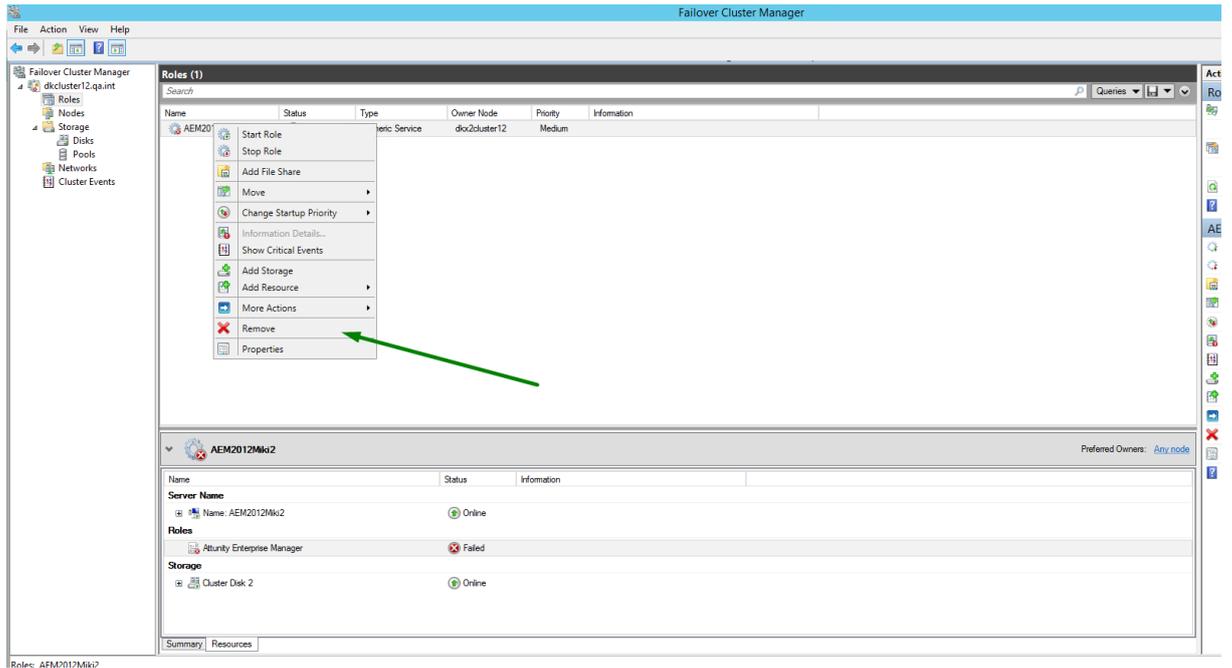
Roles (1)					
Search					
Name	Status	Type	Owner Node	Priority	Information
AEM2012Mki2	Failed	Generic Service	dlx1cluster12	Medium	

4. Go to Node 1 that should be owner to Enterprise Manager Cluster Role and Cluster Disk. Uninstall the service.
5. Move the Enterprise Manager Role to Node 2.



6. Uninstall Enterprise Manager on Node 2.
7. Remove the Enterprise Manager Cluster Role.

A Setting up High Availability



B Impact of DST change on Qlik Replicate

This topic describes the impact of Daylight Saving Time (DST) on Qlik Replicate and provides guidelines for handling changes brought about by DST.

Tasks that move from Full Load to Change Processing when DST comes into effect may encounter data loss. However, such data loss can be prevented by adhering to the guidelines outlined in this appendix.

Additionally, the times displayed in the Replicate Console may not be synced with the server. Should you encounter any time discrepancy issues, either restart the Qlik Replicate Server service or stop and resume the tasks.

There are two types of DST changes:

- **DST On** - Occurs approximately when Summer starts (actual date is country specific). Its impact on local time is that local time is moved one hour forward (so, for example, 01:00 AM becomes 02:00 AM). This DST change does not impact Qlik Replicate because it does not result in time overlap.
- **DST Off** - Occurs approximately when Winter starts (actual date is country specific). Its impact on local time is that local time is moved back one hour (so, for example, 02:00 AM becomes 01:00 AM). This DST change results in time overlap where local time travels over the same hour twice in a row.

The comments below assume that the customer has not changed the time but rather the timezone or the DST setting. Changing the actual time (not for minor time adjustments) is a sensitive operation and is best done when Qlik Replicate is stopped.

Running Qlik Replicate tasks do not depend on the timezone or DST for correctly scanning and processing the transaction logs. Internally, Qlik Replicate timers use UTC.

Still, there are several places where DST may have an effect:

1. Timestamps in logs and audit messages are in local time. As a result, when Winter time starts, the logs will show the time going back an hour; conversely, when Summer time starts, the logs may appear to be missing one hour.
2. Scheduled jobs as well as the global and table manipulation variables `timestamp` and `commit_timestamp` use local time so these will also be affected. The impact of this depends on the manipulation done and on the intended use of the timestamp based data.

B Impact of DST change on Qlik Replicate



To prevent timestamp and scheduling anomalies resulting from DST starting or ending, the following best practices should be observed:

- *DST Off (summer to winter): Do not schedule a task to start from the time the clock changes until the following hour. For example, if DST ends at 02:00 am, do not schedule a task to run between 02:00 and 02:59, as the task will run twice.*
- *DST On (winter to summer): Do not schedule a task to start from the time the clock changes until the following hour. For example, if DST starts at 02:00 am, do not schedule a task to run between 02:00 and 02:59 as this hour does not exist.*
- *The initial Full Load of tables or reloading of tables should not be done during the DST change window. It is recommended to perform such operations either an hour before or an hour after DST.*

If you have existing jobs scheduled to start at the overlap time and you do not want to modify them, then you need to stop the Qlik Replicate Server. Going in to Winter time, for example, if at 02:00 AM the clock is to be set back to 01:00 AM then when the time is 00:55 AM the Qlik Replicate Server should be stopped and, after an hour and ten minutes (at 01:05 AM), should be started again.

If you forget to do this, all scheduled jobs will run an hour earlier than intended. You can rectify this by setting the desired scheduling time and then restarting the Qlik Replicate Server service.

3. Statistics shown on the console are also sensitive to local time and thus may also show confusing/inaccurate data in the overlap period (going in to Winter time) or for the skipped period (going into Summer time).
4. If the clock on Qlik Replicate Server machine is one hour behind the clock on the Qlik Replicate Console (UI) machine, the following issues are known to occur:
 - The **Applied Changes** circle graph will be updated as the changes are applied, but the information in the **Recent Activity** tab will not be updated.
 - Scheduled jobs will start according to the Qlik Replicate Server time (as expected), but will remain in the **Active Jobs** list after execution instead of moving to the **Expired Jobs** tab.

For more information on scheduling jobs, see [Scheduling jobs \(page 266\)](#).

In general, it is recommended to avoid non-critical task design changes during the first overlap period (going in to Winter time) so as to prevent confusion about when the changes took place.

In addition to Qlik Replicate, other components are also involved including:

- The source endpoint system
- The target endpoint system
- The local operating system
- The task design (specifically using timestamp based variables)

Given the complexity of the topic and the involvement of many independent components and settings, Qlik generally recommends that customers first verify the impact of DST changes in their test environment.