1 What's new? ................................................................. 8
  1.1 What's new in Data Warehouse projects? ............................. 8
    Uniform source consolidation .................................. 8
    Environment variables ............................................. 8
    Snowflake VARIANT data type support ............................ 9
    Support for data profiling and data quality rules when using Google Cloud BigQuery ............................. 9
    Attributes case sensitivity support ................................ 9
    Associating a Replicate task that writes to a Hadoop target ................ 9
    Performance improvements ..................................... 10
    Data mart UX improvement .................................... 10
  1.2 What's new in Data Lake projects? .................................. 10
    Support for excluding deleted records from ODS views .......... 10
    Databricks on Google Cloud .................................... 11
    Generating projects using the CLI ................................ 11
    The "Associate with Replicate task option" is now mandatory ...... 11
  1.3 All Projects .......................................................... 11
    Task Recovery ...................................................... 11
    New Project title setting ...................................... 13
    Support for multiple Replicate Servers .......................... 13
    License ............................................................ 13
    Performance ....................................................... 14
  1.4 Support for Microsoft Edge Browser ................................ 14
  1.5 Compose Support in Enterprise Manager ............................ 14

2 Introduction .............................................................. 15
  2.1 Data warehouse projects ........................................... 15
    Data warehouse projects architecture ............................ 15
    Key features ....................................................... 16
  2.2 Data lake projects .................................................. 16
    Easy data structuring and transformation ......................... 16
    Continuous updates ............................................. 16
    Historical data store ............................................ 16
    Data lake project architecture .................................. 17

3 Qlik Compose installation and setup ...................................... 18
  3.1 Preparing your system for Compose ................................ 18
    Hardware prerequisites .......................................... 18
    Software and network prerequisites ............................. 19
    Required permissions for the Compose service .................... 19
    Reserved system names ........................................ 19
  3.2 Installing or upgrading Compose ................................... 19
    Installation Instructions ......................................... 19
    Upgrade Instructions ........................................... 20
  3.3 Installing and upgrading Compose silently ............................ 20
    Silently installing Compose .................................... 21
    Silently upgrading Compose ................................... 22
    Silently uninstalling Compose .................................. 22
  3.4 Determining the required number of database connections .............. 22
  3.5 Accessing Qlik Compose ............................................ 23
## 4 Security considerations

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.1 Setting up HTTPS for the Compose console</td>
<td>25</td>
</tr>
<tr>
<td>Checking if an SSL certificate is installed</td>
<td>25</td>
</tr>
<tr>
<td>Using the self-signed certificate</td>
<td>26</td>
</tr>
<tr>
<td>Replacing the self-signed certificate on Windows</td>
<td>28</td>
</tr>
<tr>
<td>4.2 Setting the hostname and changing the HTTPS port</td>
<td>29</td>
</tr>
<tr>
<td>4.3 Setting up HSTS on Compose</td>
<td>29</td>
</tr>
<tr>
<td>Enabling HSTS</td>
<td>30</td>
</tr>
<tr>
<td>Disabling HSTS</td>
<td>30</td>
</tr>
<tr>
<td>4.4 Setting Single Sign-On Authentication with Kerberos</td>
<td>30</td>
</tr>
<tr>
<td>4.5 Changing the master user password</td>
<td>31</td>
</tr>
</tbody>
</table>

## 5 Data Warehouse projects

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.1 Defining a Qlik Replicate task</td>
<td>34</td>
</tr>
<tr>
<td>Prerequisites</td>
<td>34</td>
</tr>
<tr>
<td>Limitations and considerations</td>
<td>34</td>
</tr>
<tr>
<td>Setting up the task</td>
<td>35</td>
</tr>
<tr>
<td>5.2 Adding and managing data warehouse projects</td>
<td>35</td>
</tr>
<tr>
<td>Adding data warehouse projects</td>
<td>36</td>
</tr>
<tr>
<td>Managing and monitoring projects</td>
<td>37</td>
</tr>
<tr>
<td>Project settings</td>
<td>38</td>
</tr>
<tr>
<td>Resetting projects</td>
<td>44</td>
</tr>
<tr>
<td>Project deployment</td>
<td>45</td>
</tr>
<tr>
<td>Migrating objects as CSV files</td>
<td>46</td>
</tr>
<tr>
<td>Exporting and importing projects using the CLI</td>
<td>75</td>
</tr>
<tr>
<td>Working with environment variables</td>
<td>82</td>
</tr>
<tr>
<td>Generating projects using the CLI</td>
<td>92</td>
</tr>
<tr>
<td>Exporting project documentation</td>
<td>93</td>
</tr>
<tr>
<td>Viewing and downloading DDL scripts</td>
<td>94</td>
</tr>
<tr>
<td>Project versioning</td>
<td>95</td>
</tr>
<tr>
<td>Creating a diagnostics package</td>
<td>96</td>
</tr>
<tr>
<td>5.3 Getting started with Data Warehouse projects</td>
<td>97</td>
</tr>
<tr>
<td>High-level flow</td>
<td>97</td>
</tr>
<tr>
<td>Console elements</td>
<td>97</td>
</tr>
<tr>
<td>Data warehouse project tutorial</td>
<td>100</td>
</tr>
<tr>
<td>5.4 Setting up a data warehouse connection</td>
<td>104</td>
</tr>
<tr>
<td>Using Microsoft SQL Server as a data warehouse</td>
<td>105</td>
</tr>
<tr>
<td>Using Oracle as a data warehouse</td>
<td>108</td>
</tr>
<tr>
<td>Using Snowflake as a data warehouse</td>
<td>111</td>
</tr>
<tr>
<td>Using Amazon Redshift as a data warehouse</td>
<td>115</td>
</tr>
<tr>
<td>Using Microsoft Azure Synapse Analytics as a data warehouse</td>
<td>117</td>
</tr>
<tr>
<td>Using Google Cloud BigQuery as a Data Warehouse</td>
<td>120</td>
</tr>
<tr>
<td>Managing databases</td>
<td>123</td>
</tr>
<tr>
<td>5.5 Setting up Landing Zone and Data Source connections</td>
<td>124</td>
</tr>
<tr>
<td>Permissions</td>
<td>124</td>
</tr>
<tr>
<td>Data type mappings</td>
<td>125</td>
</tr>
<tr>
<td>Defining landing zones</td>
<td>132</td>
</tr>
<tr>
<td>Defining Replicate data source connections</td>
<td>134</td>
</tr>
</tbody>
</table>
## Contents

Managing databases ................................................................. 141
5.6 Creating and managing the model ........................................ 141
Generating the model ............................................................. 142
Model limitations ................................................................. 149
Validating the model ............................................................. 149
Displaying the model ............................................................. 150
Managing the model ............................................................. 153
Creating expressions ............................................................. 168
Opening the expression builder ............................................. 169
Defining reusable transformations ......................................... 176
5.7 Creating and managing the data warehouse ......................... 177
Data warehouse tasks ............................................................. 178
Managing tasks ................................................................. 187
Viewing and exporting task statements ................................... 202
Modifying task settings ......................................................... 203
Validating the data warehouse ................................................. 208
Clearing the data warehouse metadata cache ......................... 210
5.8 Creating and managing data marts ....................................... 211
Adding data marts and star schemas ....................................... 211
Displaying data in a pivot table ............................................... 217
 Managing data marts ............................................................. 219
Example of a Valid Table Creation Modifier ............................ 225
Example of a Valid Table Creation Modifier ............................ 230
Pre and Post Loading ETLs ..................................................... 233
Viewing and exporting task statements ................................... 234
Validating the data mart ......................................................... 235
Modifying data mart settings ............................................... 236
The "Obsolete" indicator ......................................................... 238
5.9 Creating and managing command tasks ............................... 238
Defining command tasks ....................................................... 239
Managing command tasks ..................................................... 239
Controlling and monitoring command tasks ............................ 239
5.10 Controlling and monitoring tasks and workflows ............... 240
Viewing information in the monitor ......................................... 240
Viewing missing references .................................................... 242
Controlling tasks ................................................................. 244
Notifications ............................................................................ 246
Workflows .............................................................................. 248
Monitoring and controlling Qlik Replicate tasks ..................... 252
6 Data Lake projects .................................................................. 255
6.1 Defining a Qlik Replicate task ............................................ 255
Prerequisites ............................................................................. 255
Limitations and Considerations .............................................. 256
Setting up the task ................................................................. 256
6.2 Adding and managing Data Lake projects ......................... 257
Prerequisites ............................................................................. 257
Data Lake project guidelines .................................................. 259
## Contents

- Adding data lake projects ................................................................. 260
- Managing and monitoring projects ...................................................... 262
- Project settings .................................................................................. 263
- Resetting projects .............................................................................. 268
- Project deployment .............................................................................. 269
- Exporting and importing projects using the CLI .................................... 270
- Generating projects using the CLI ......................................................... 276
- Viewing and downloading DDL scripts ................................................. 277
- Project versioning ............................................................................... 278
- Creating a diagnostics package ............................................................. 280

### 6.3 Getting started with Data Lake projects ........................................... 280
- High-level flow ................................................................................... 281
- Console elements ................................................................................ 281

### 6.4 Setting up landing and storage connections ....................................... 283
- Defining a Storage Zone ....................................................................... 284
- Defining Landing Zones ..................................................................... 292
- Managing Landing and Storage connections ........................................ 294

### 6.5 Selecting source tables and managing metadata ................................ 294
- Selecting and adding the source tables ................................................ 294
- Validating the metadata and storage ..................................................... 298
- Managing the metadata ....................................................................... 300
- Schema evolution ................................................................................. 304
- Creating transformations ..................................................................... 307
- Reusable transformations .................................................................... 313

### 6.6 Creating and Managing Storage Zone Tasks ...................................... 315
- Defining and running data storage tasks ............................................... 316
- Managing task definitions .................................................................... 319
- Clearing the metadata cache ................................................................ 328
- Viewing and exporting task statements ............................................... 329
- Modifying task settings ....................................................................... 329

### 6.7 Creating and managing command tasks ............................................ 330
- Defining Command tasks ..................................................................... 330
- Managing Command tasks ................................................................... 331
- Controlling and monitoring Command tasks ........................................ 331

### 6.8 Controlling and monitoring tasks and workflows ............................... 331
- Viewing information in the monitor ...................................................... 332
- Running and controlling tasks .............................................................. 333
- Notifications ...................................................................................... 335
- Workflows ......................................................................................... 337
- Monitoring and controlling Replicate tasks .......................................... 341

### 7 Managing Compose ............................................................................. 343

#### 7.1 License settings ............................................................................. 343
- License enforcement ............................................................................. 343
- Registering a license ........................................................................... 343

#### 7.2 Viewing a license .......................................................................... 344

#### 7.3 Logging settings ............................................................................ 344
- Setting the logging level ...................................................................... 344
## Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Setting automatic roll over and cleanup</td>
<td>345</td>
</tr>
<tr>
<td>Viewing and downloading Compose log files</td>
<td>346</td>
</tr>
<tr>
<td>7.4 Mail server settings</td>
<td>347</td>
</tr>
<tr>
<td>7.5 Running tasks on a remote Compose server</td>
<td>347</td>
</tr>
<tr>
<td>7.6 Replicate Server settings</td>
<td>347</td>
</tr>
<tr>
<td>7.7 User permissions</td>
<td>348</td>
</tr>
<tr>
<td>Default user permissions according to role</td>
<td>349</td>
</tr>
<tr>
<td>Granular access control</td>
<td>350</td>
</tr>
<tr>
<td>Managing user permissions</td>
<td>352</td>
</tr>
<tr>
<td>7.8 Audit trails</td>
<td>355</td>
</tr>
<tr>
<td>Audit trail information</td>
<td>355</td>
</tr>
<tr>
<td>Exporting Audit Trail files</td>
<td>356</td>
</tr>
<tr>
<td>Configuring Audit Trail size and retention</td>
<td>357</td>
</tr>
<tr>
<td>Decoding an encoded payload</td>
<td>357</td>
</tr>
<tr>
<td>8 Setting up Compose on a Windows HA cluster</td>
<td>358</td>
</tr>
<tr>
<td>8.1 Step 1: Installing Compose in the cluster</td>
<td>358</td>
</tr>
<tr>
<td>Preparation</td>
<td>358</td>
</tr>
<tr>
<td>Primary node setup</td>
<td>358</td>
</tr>
<tr>
<td>Secondary node setup</td>
<td>359</td>
</tr>
<tr>
<td>8.2 Step 2: Adding the Compose service</td>
<td>359</td>
</tr>
<tr>
<td>8.3 Step 3: Defining the service dependencies</td>
<td>360</td>
</tr>
<tr>
<td>8.4 Step 4: Defining the URL for the cluster</td>
<td>360</td>
</tr>
<tr>
<td>8.5 Upgrading Compose on the cluster</td>
<td>361</td>
</tr>
<tr>
<td>A Impact of DST change on Qlik Compose</td>
<td>362</td>
</tr>
<tr>
<td>B Support matrix</td>
<td>363</td>
</tr>
<tr>
<td>B.1 Supported Windows platforms</td>
<td>363</td>
</tr>
<tr>
<td>B.2 Supported browsers</td>
<td>363</td>
</tr>
<tr>
<td>B.3 Supported Qlik Replicate and Enterprise Manager versions</td>
<td>363</td>
</tr>
<tr>
<td>B.4 Supported Databases for Data Warehouse Projects</td>
<td>364</td>
</tr>
<tr>
<td>Supported data sources</td>
<td>364</td>
</tr>
<tr>
<td>Supported data warehouses</td>
<td>364</td>
</tr>
<tr>
<td>B.5 Supported databases for Data Lake projects</td>
<td>366</td>
</tr>
<tr>
<td>C Cron format and examples</td>
<td>367</td>
</tr>
<tr>
<td>C.1 Cron format</td>
<td>367</td>
</tr>
<tr>
<td>C.2 Special characters</td>
<td>367</td>
</tr>
<tr>
<td>C.3 Usage examples</td>
<td>368</td>
</tr>
<tr>
<td>D Supported characters</td>
<td>370</td>
</tr>
<tr>
<td>E Glossary</td>
<td>371</td>
</tr>
</tbody>
</table>
1  What's new?

The following section describes the enhancements and new features introduced in Qlik Compose August 2021.

1.1  What's new in Data Warehouse projects?

The following section describes the enhancements and new features introduced in Qlik Compose Data Warehouse projects.

Uniform source consolidation

_requires Compose August 2021 Service Release 03 or later._

To enable uniform source consolidation configuration, a new **Consolidation** tab has been added to the data warehouse task settings.

When the **Consolidate uniform sources** option is enabled, Compose will read from the selected data sources and write the data to one consolidated entity. This is especially useful if your source data is managed across several databases with the same structure, as instead of having to define multiple data warehouse tasks (one for each source), you only need to define a single task that consolidates the data from the selected data sources.

**Consolidation tab showing selected data sources**

Environment variables

_requires Compose August 2021 Service Release 03 or later._
Environment variables allow developers to build more portable expressions, custom ETLs, and Compose configurations, which is especially useful when working with several environments, as is the case with DTAP (Development, Testing, Acceptance and Production). Different environments (for example, development and production) often have environment-specific settings such as database names, schema names, and Replicate task names. Variables allow you to easily move projects between different environments without needing to manually configure the settings for each environment. This is especially useful if many settings are different between environments. For each project, you can use the predefined environment variables or create your own environment variables.

**Snowflake VARIANT data type support**

When creating a Landing Zone, the new **Discover the VARIANT data type as** option lets you choose whether columns of the VARIANT data type will be created as JSON or XML in the Snowflake database.

> **VARIANT data type support in Replicate is planned for the November 2021 release. Until then, this feature can be used to discover tables with this data type that were created manually in the Landing Zone, or to manually set certain columns in the model to the JSON/XML data type.**

**Support for data profiling and data quality rules when using Google Cloud BigQuery**

> **Requires Compose August 2021 Service Release 01 or later.**

Added support for data profiling and data quality rules when using Google Cloud BigQuery as a data warehouse.

**Attributes case sensitivity support**

> **Requires Compose August 2021 Service Release 01 or later.**

In previous versions, attempting to create several Attributes with the same name but a different case would result in a duplication error. Now, such attributes will be created with an integer suffix that increases incrementally for each attribute added with the same name. For example: Sales, SALES_01, and Sales_02.

**Associating a Replicate task that writes to a Hadoop target**

> **Requires Compose August 2021 Service Release 01 or later.**

This version introduces support for associating a Replicate task that writes to a Hadoop target.
1 What's new?

Performance improvements

- The "Discover" queries have been optimized to return results much faster than in previous versions.
- When loading the data mart tables in Microsoft Azure Synapse Analytics, the 'FORCE ORDER' hint has been added to the statements that populate the temporary DIM_MUT tables and to the statements that populate the fact tables. For each dimension, the statistics are updated the moment before they are used to load the fact tables.
- The New Star Schema wizard has been scaled to allow a significantly larger number of tables to be selected.

Performance improvements that require Compose August 2021 Service Release 01 or later

- Validating a model with self-referencing entities is now significantly faster than in previous versions. For instance, it now takes less than a minute (instead of up to two hours) to validate a model with 5500 entities.
- The time it takes to "Adjust" the data warehouse has been significantly reduced. For instance, it now takes less than three minutes (instead of up to two hours) to adjust a data warehouse with 5500 entities.
- Significantly improved the loading speed of data mart Type 2 dimensions with more than two entities. In order to benefit from this improvement, customers upgrading with existing data marts needs to regenerate their data mart ETLs.

Data mart UX improvement

**Requires Compose August 2021 Service Release 01 or later.**

The Data Mart Dimensions tree and the Star Schema Fact tab were redesigned to provide a better user experience.

1.2 What's new in Data Lake projects?

The following section describes the enhancements and new features introduced in Qlik Compose Data Lake projects.

Support for excluding deleted records from ODS views

**Requires Compose August 2021 Service Release 03 or later.**

A Deleted records in ODS views section has been added to the General tab of the project settings, with the following options:

- **Exclude the corresponding record from the ODS views** - This is the default option as records marked as deleted should not usually be included in ODS views.

Setup and User Guide - Qlik Compose, August 2021
What's new?

- **Include the corresponding record in the ODS views** - Although not common, in some cases, you might want include records marked as deleted in the ODS views in order to analyze the number of deleted records and investigate the reason for their deletion. Also, regulatory compliance might require you to be able to retrieve the past record status (which requires change history as well).

  As this was the default behavior in previous versions, you might need to select this option to maintain backward compatibility.

Databricks on Google Cloud

*Requires Replicate May 2021 Patch Release 02 or later.*

Databricks on Google Cloud 8.x is now supported.

Generating projects using the CLI

Previously available only for Data Warehouse projects, this version introduces support for generating Data Lake projects using the Compose CLI. This can be especially useful when deploying projects between different environments. When the `generate_project` command is run, Compose:

- Validates the metadata.
- Creates any storage tables that do not exist.
- Validates the storage.
- Adjusts the storage if needed.
- Generates all storage tasks.

The "Associate with Replicate task option" is now mandatory

When defining a Landing Zone for Data Lake projects, associating a Replicate task (by selecting the Associate with Replicate task check box) is now mandatory. Therefore, after upgrading to this version, make sure to set this option if it is not already set.

1.3 All Projects

The following section describes the enhancements and new features that are common to both Data Lake projects and Data warehouse projects.

Task Recovery

The ability to set SQL state classes and error codes for triggering task recovery has been added to the project settings' Environment tab.
Limitations and considerations:

- Schema evolution retries are not supported.
- ODBC statements comprise a small part of the task execution sequence. However, as the task retry mechanism is JDBC-based, ODBC statements will not be retried even if the specified SQL state/error code is encountered.
New Project title setting

A new Project title setting had been added to the Environment tab of the project settings. The project title will be shown in the console banner. If both an Environment Title and a Project Title are defined, the project title will be displayed to the right of the environment title. Unlike the Environment title and Environment type, which are unique for each environment, the project title is environment independent. This means that the project title will always be retained, even when deploying to a different environment.

The following image shows the banner with both an Environment title and a Project title:

![Banner with both Environment title and Project title](image)

The banner text is shown without the Environment title and Project title console labels. This provides greater flexibility as it allows you add any banner text you like, regardless of the actual label name. For example, specifying Project owner: Mike Smith in the Project title field, will display that text in the banner.

Support for multiple Replicate Servers

This version introduces support for configuring connectivity to Multiple Replicate Servers. This can be especially useful if you need to set up several projects in Compose, each with a different Replicate Server and task.

License

The parameter for registering a license has been changed from --req to --infile, thereby aligning it with other Qlik data integration products. In addition, the need to append the path with an "@" symbol is no longer applicable.
1  What's new?

Example of previous command:

ComposeCLI.exe register_license --req @c:\Admin\Temp\lic.txt

Example of new command:

ComposeCLI.exe register_license --infile c:\Admin\Temp\lic.txt

Performance
Creating and dropping tables via Compose is now significantly faster when a large number of tables is involved.

1.4  Support for Microsoft Edge Browser

Requires Compose August 2021 Service Release 01 or later.

This version introduces support for accessing the Compose console using Microsoft Edge.

1.5  Compose Support in Enterprise Manager

From Enterprise Manager Service Release SR2, the following functionality will now be supported when working with Qlik Compose August 2021 (or above):

- Monitoring of Data Lake projects
- Monitoring of workflows - Note that monitoring information is provided at the workflow level only. For information on individual tasks in the workflow, you need to drill-down to the Compose project from Enterprise Manager.
- Monitoring tasks that have the same name, but are defined in different Compose projects. To facilitate such support:
  - In the API, Compose task names are now appended with a double underscore and the project name. For example, MyTask__MyProject.

Task names without an appended project name are no longer supported. Therefore, any scripts or applications using the old name format (i.e. without the __projectName suffix) will need to be updated to use the new format.

- In the console user interface, a new Project column has been added, allowing you to easily differentiate between two tasks with the same name.

The newly supported functionality has also been incorporated into the following API methods (REST, .NET, and Python): GetTaskList, GetTaskDetails, RunTask, and StopTask.
2 Introduction

Qlik Compose provides an all-in-one purpose built automation solution for creating an agile data warehouse and/or ingesting data from multiple sources to your data lake for further downstream processing. To this end, Qlik Compose offers two project types: Data Warehouse and Data Lake. This introduction will take a closer look at how these projects can help your organization overcome the hurdles typically faced when confronted with the challenge of setting up and maintaining an agile data warehouse, or when faced with challenge of ingesting data from multiple source to a single analytics-ready storage system.

2.1 Data warehouse projects

Traditional methods of designing, developing, and implementing data warehouses require large time and resource investments. The ETL stand-up development effort alone - multi-month and error-prone with prep times of up to 80 percent and expertise from specialized developers - often means your data model is out of date before your BI project even starts. Plus, the result of a traditional data warehouse design, development, and implementation process is often a system that can’t adapt to continually changing business requirements. Yet modifying your data warehouse diverts skilled resources from your more innovation-related projects. Consequently, your business ends up with your data warehouse becoming a bottleneck as much as an enabler of analytics.

Qlik Compose data warehouse projects allows you to automate these traditionally manual, repetitive data warehouse tasks: design, development, testing, deployment, operations, impact analysis, and change management. Qlik Compose automatically generates the task statements, data warehouse structures, and documentation your team needs to efficiently execute projects while tracking data lineage and ensuring integrity. Using Qlik Compose, your IT teams can respond fast - in days - to new business requests, providing accurate time, cost, and resource estimates. Then once projects are approved, your IT staff can finally deliver completed data warehouses, data marts, and BI environments in far less time.

Data warehouse projects architecture

The process is illustrated in the following diagram and described below:
Key features

The comprehensive set of automation features in our Qlik Compose solution simplifies data warehousing projects. It eliminates the cumbersome and error-prone manual coding required by legacy data warehouse design and implementations’ many repetitive steps. In addition, our solution includes the operational features your business needs for ongoing data warehouse and data mart maintenance.

**Automation Features**
- Optimized for either model-driven or data-driven data warehousing approaches
- Real-time source data integration
- Automated ETL generation
- Physical data warehouse management
- Data mart generation

**Operational Features**
- Monitoring
- Workflow designer and scheduler
- Notifications
- Data profiling and quality enforcement
- Lineage and impact analysis
- Project documentation generation
- Migration between environments

2.2 Data lake projects

Leverage Qlik Compose data lake projects to automate your data pipelines and create analytics-ready data sets. By automating data ingestion, schema creation, and continual updates, organizations realize faster time-to-value from their existing data lake investments.

Easy data structuring and transformation

An intuitive and guided user interface helps you build, model and execute data lake pipelines. Automatically generate schemas and Hive Catalog structures for operational data stores (ODS) and historical data stores (HDS) without manual coding.

Continuous updates

Be confident that your ODS and HDS accurately represent your source systems.

- Use change data capture (CDC) to enable real-time analytics with less administrative and processing overhead.
- Efficiently process initial loading with parallel threading.
- Leverage time-based partitioning with transactional consistency to ensure that only transactions completed within a specified time are processed.

Historical data store

Derive analytics-specific data sets from a full historical data store (HDS).

- New rows are automatically appended to HDS as data updates arrive from source systems.
- New HDS records are automatically time-stamped, enabling the creation of trend analysis and other time-oriented analytic data marts.
- Supports data models that include Type-2, slowing changing dimensions.
Data lake project architecture

The flow is as follows:

1. **Land**: The source tables are loaded into the Landing Zone using Qlik Replicate or other third-party replication tools. When using Qlik Replicate to move the source table to the Landing Zone, you can define either a Full Load replication task or a Full Load and Store Changes task to constantly propagate the source table changes to the Landing Zone in write-optimized format.

2. **Store**: After the source tables are present in the Landing Zone, Compose auto-generates metadata based on the data source(s). Once the metadata and the mappings between the tables in the Landing Zone and the Storage Zone have been finalized, Compose creates and populates the Storage Zone tables in read-optimized format, ready for consumption by downstream applications.

It should be noted that even though setting up the initial project involves both manual and automatic operations, once the project is set up, you can automate the tasks by designing a Workflow in Compose and/or utilizing the Compose scheduler.
3 Qlik Compose installation and setup

This section describes how to install and set up Qlik Compose.

Note that as Qlik Replicate serves as a data (and metadata) provider for Qlik Compose, you also need to install Replicate in your organization. For a description of the Replicate installation procedure, refer to the Qlik Replicate Setup and User Guide.

In this section:

- Preparing your system for Compose (page 18)
- Installing or upgrading Compose (page 19)
- Installing and upgrading Compose silently (page 20)
- Determining the required number of database connections (page 22)
- Accessing Qlik Compose (page 23)

3.1 Preparing your system for Compose

Compose should be installed on a Windows Server machine that is able to access the data warehouse and optionally the source database(s) defined in your Compose project. Note that Compose only needs to access the source database if you plan to discover the source database when generating your model. For more information on discovery, see Discovering the Source Database or Landing Zone (page 142).

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

Hardware prerequisites

The following table lists the required hardware for varied deployment scales:

<table>
<thead>
<tr>
<th>Component</th>
<th>Basic System</th>
<th>Large System</th>
<th>Extra-Large System</th>
</tr>
</thead>
<tbody>
<tr>
<td>Processor</td>
<td>Quad core</td>
<td>Quad core base</td>
<td>8-core base</td>
</tr>
<tr>
<td>Additional cores may improve performance when several ETL processes are running concurrently.</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Memory</td>
<td>8 GB</td>
<td>16 GB</td>
<td>32 GB</td>
</tr>
<tr>
<td>Additional memory may improve performance when several ETL processes are running concurrently.</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Component

<table>
<thead>
<tr>
<th>Disk requirements</th>
<th>Basic System</th>
<th>Large System</th>
<th>Extra-Large System</th>
</tr>
</thead>
<tbody>
<tr>
<td>SSD</td>
<td>100 GB</td>
<td>500 GB</td>
<td>500 GB</td>
</tr>
<tr>
<td>10,000 RPM RAID</td>
<td></td>
<td>15,000 RPM RAID</td>
<td></td>
</tr>
</tbody>
</table>

*For all configurations, RAID is recommended for higher system availability in case of disk failure.*

| Network           | 1 Gb         | 10 Gb        | Two 10 Gb         |

### Software and network prerequisites

- Firewall ports 80/443 should be open on the Compose machine.
- Microsoft Visual Studio C++ 2010 X64 Redistributable installed on the Compose machine.
- .NET Framework 4.8 or later installed on the Compose machine.
- 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.*

For information on supported databases and browsers, see *Support matrix (page 363).*

### Required permissions for the Compose service

Qlik Compose needs to be installed and run as Administrator.

### Reserved system names

All database object names (queries, tables, columns, schemas, and indexes) starting with the prefix `qlk_ _`, and regardless of case, are reserved for internal Compose use.

Thus, a table named `qlK__MyTable` or a column named `QLK__MyColumn` would not be permitted.

### 3.2 Installing or upgrading Compose

The following topic describes how to install and upgrade Qlik Compose.

### Installation Instructions

*For best performance when using cloud-based databases (such as, Snowflake) as your data source or data warehouse, it is strongly recommended to install Qlik Compose on a machine (such as Amazon EC2) located in the same region as your database instance.*
To install Compose:

1. Run the Compose setup file (`Qlik_Compose_<version.number>.exe`). The Qlik Compose setup wizard opens.
2. Click **Next**. Select I accept the terms of the license agreement and then click **Next** again.
3. Optionally change the installation directory and then click **Next**.
4. Click **Next** and then click **Next** again to start the installation.
5. When the installation completes, click **Finish** to exit the Wizard.

   **As part of the installation, a new Windows Service called Qlik Compose is created.**

6. Open the Qlik Compose console as described in Accessing Qlik Compose (page 23).

   **When you first open the Qlik Compose Console, you will be prompted to register an appropriate license. Register the license that you received from Qlik.**

Upgrade Instructions

**Depending on your existing Compose version, you may also need to perform additional version-specific upgrade tasks. It is therefore strongly recommended to review the release notes for the new version before upgrading.**

1. Stop all Compose tasks and services.
2. After the Qlik Compose service has been stopped by the Installer, make sure that all child processes are also stopped.

   **Compose runs a check to verify the termination of tasks and processes before running an upgrade. If any processes are found to be still running, the installation will be aborted.**

3. Run the Qlik Compose setup wizard.
4. Start all Compose tasks and services.

### 3.3 Installing and upgrading Compose silently

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

**Before commencing the installation, make sure that the prerequisites have been met. See Preparing your system for Compose (page 18).**

The following topics describe how silently install, upgrade, and uninstall Compose:
Silently installing Compose

The installation process consists of two stages: creating a response file, and running the silent install.

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 Compose setup file, run the following command (note that this will also install Compose):
   ```
   Qlik_Compose_<version.number>.exe /r /f1<my_response_file>
   where:
   <my_response_file> is the full path to the generated response file.
   Example:
   Qlik_Compose_<version.number>.exe /r /f1:C:\Compose_install.iss
   ```

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. **Compose_install_64.iss**.

Running the silent install

To silently install Compose, open a command prompt and change the working directory to the directory containing the Compose setup file. Then issue the following command (where `<response file>` is the path to the response file you created earlier):

**Syntax:**

```
<Compose_setup_file> /s /f1<my_response_file> [/f2<LOG_FILE>]
```

**Example:**

```
C:\>Qlik_Compose_<version.number>.exe /s /f1:C:\temp\1\Compose_install.iss /f2:C:\temp\1\silent_x64_install.log
```

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

```
[ResponseResult]
ResultCode=0
```
Silently upgrading Compose

Before starting the silent upgrade:

1. Create a response file. See Step 1 of “Creating a Response File” in Silently installing Compose (page 21)
2. It is strongly recommended to back up the Compose "Data" folder.
3. All tasks and java processes must be terminated. Compose runs a check to verify the termination of tasks and processes before running the upgrade. If any processes are found to be still running, the upgrade will be aborted.

To silently upgrade Compose:

1. Open a command prompt and change the working directory to the directory containing the Compose setup file.
2. Issue the following command (where <my_response_file> is the path to the response file you created earlier):

Syntax:

```
<COMPOSE_KIT> /s /f1<my_response_file> [/f2<LOG_FILE>]
```

Example:

```
C:\>Qlik_Compose_<version.number>.exe /s /f1:C:\temp\1\Compose_upgrade.iss /f2:C:\temp\1\silent_x64_up.log
```

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

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

Silently uninstalling Compose

Silently uninstalling Compose is also comprised of creating a response file and running the silent uninstall.

The process is the same as for silently installing Compose. For instructions, see Silently installing Compose (page 21)

3.4 Determining the required number of database connections

As a rule of thumb, the higher the number of database connections opened for Compose, the more tables Compose will be able to load in parallel. It is therefore recommended to open as many database connections as possible for Compose. However, if the number of database connections that can be opened for Compose is limited, you can calculate the minimum number of required connections as described below.
3 Qlik Compose installation and setup

To determine the number of required connections:

1. For each task, determine the number of connections it can use during runtime. This value should be specified in the Advanced tab in the Manage Data Warehouse Tasks Settings window (Data Warehouse projects) or in the Manage Storage Tasks Settings window (Data Lake projects). When determining the number of required connections, various factors need to be taken into account including the number of tables, the size of the tables, and the volume of data. It is therefore recommended to determine the required number of connections in a Test environment.

2. Calculate the number of connections needed by all tasks that run in parallel. For example, in a Data Lake project, if three data storage tasks run in parallel, and each task requires 5 connections, then the number of required connections will be 15.

   Similarly, in a Data Warehouse project, if a workflow contains two data warehouse tasks that run in parallel and each task requires 5 connections, then the minimum number of required connections will be 10. However, if the same workflow also contains two data mart tasks (that run in parallel) and the sum of their connections is 20, then the minimum number of required connections will be 20.

3. Factor in the connections required by the Compose Console. To do this, multiply the maximum number of concurrent Compose users by three and then add to the sum of Step 2 above. So, if the number of required connections is 20 and the number of concurrent Compose users is 4, then the total would be:
   \[ 20 + 12 = 32 \]

3.5 Accessing Qlik Compose

You can use a Web browser to access the Qlik Compose Console from any computer in your network. For information on supported browsers, see Preparing your system for Compose (page 18).

The person logged in to the computer where you are accessing the Console must be an authorized Qlik Compose user. For more information, see Managing user permissions (page 352).

To access the Qlik Compose Console:

1. To access the Qlik Compose Console from the machine on which it is installed, select All Programs > Qlik Compose > Qlik Compose Console from the Windows Start menu. To access the Qlik Compose Console from a remote browser, type the following address in the address bar of your Web browser
   
   https://<ComputerName>/qlikcompose/
   
   Where <ComputerName> is the name or IP address of the computer on which Compose is installed.

2. If no server certificate is installed on the Compose machine, a page stating that the connection is untrusted will be displayed. This is because when Compose detects that no server certificate is installed, it installs a self-signed certificate. Since 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
3 Qlik Compose installation and setup

*Compose console (page 25).*

3. When prompted for your password, enter your domain username and password.
4 Security considerations

During normal operation, Qlik Compose needs to access databases and storage systems for the purpose of reading and writing data and metadata.

This section describes the procedure you should follow to ensure that any data handled by Qlik Compose will be completely secure.

In this section:

- Setting up HTTPS for the Compose console (page 25)
- Setting the hostname and changing the HTTPS port (page 29)
- Setting up HSTS on Compose (page 29)
- Setting Single Sign-On Authentication with Kerberos (page 30)
- Changing the master user password (page 31)

4.1 Setting up HTTPS for the Compose console

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

As Compose 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 Compose 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:

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

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

```bash
netsh http show sslcert | findstr /c:"^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 Qlik Compose web user interface will automatically be available for secure access from a web browser using the following URL:

```
https://<ComputerName>/qlikcompose/
```

Where `<ComputerName>` is the name or IP address of the computer on which Compose is installed.
Using the self-signed certificate

Due to the way the HTTPS protocol works, there is no way for Compose to automatically provide and install a valid SSL server certificate. Still, in the event that no SSL server certificate is installed, Compose automatically generates and installs a self-signed SSL server certificate (as a temporary measure). This certificate is generated on the Compose 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:

![Chrome warning page](https://example.com/your_connection_is_not_private.png)

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:

```
composeCtl.exe certificate clean
```

Note that after the self-signed certificate is deleted, connections to the Qlik Compose 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 Qlik Compose service.
4 Security considerations

Replacing the self-signed certificate on Windows

The instructions below are intended for organizations who wish to replace the self-signed certificate generated by the Compose Server on Windows with their own certificate. The process, which is described below, involves removing the self-signed certificate and then importing the new certificate.

See also Setting up HTTPS for the Compose console (page 25).

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 name browsers will use to access the machine.

To remove the self-signed certificate created by Qlik Compose:

1. Stop the Qlik Compose service.
2. Open a command prompt (using the "Run as administrator" option) and change the path to the Compose bin directory. The default path is C:\Program Files\Qlik\Compose\bin.
3. Run the following command:
   ```
   composectl.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 window 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 window.
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 window 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 window.
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 window 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 the following commands:
4 Security considerations

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

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

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

Example:
```bash
   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 Compose service.

4.2 Setting the hostname and changing the HTTPS port

After installing Qlik Compose, you can use the Compose CLI to set the hostname and HTTPS port for accessing the Qlik Compose 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 Compose machine by running the command shown below.

To set the hostname:

Run the following command:
```bash
<product_dir>/bin\ComposeCtl.exe configuration set --address address
```
where `address` is the hostname of the Compose server machine.

To change the HTTPS port:

Run the following command:
```bash
<product_dir>/bin\ComposeCtl.exe configuration set --https_port port_number
```
where `port_number` is the HTTPS port number of the Compose server machine. The default HTTPS port is 443.

4.3 Setting up HSTS on Compose

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 Compose Web UI and/or the Compose 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

**Syntax:**

```plaintext
ComposeCtl.exe configuration set --static_http_headers header_list --rest_http_headers header_list
```

where `--static_http_headers` are the headers required to connect to the Compose Web UI and `--rest_http_headers` are the headers required to connect using the API.

Headers should be specified using the following format:

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

**Example:**

The following instructs the browser to treat the domain as an HSTS host for a year (there are approximately 31536000 seconds in a year). The optional `includeSubDomains` directive means that subdomains (i.e. `secure.myhealthcare.example.com`) should also be treated as an HSTS domain.

```plaintext
ComposeCtl.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 revert to regular HTTPS connections by running the following command:

To disable static_http_headers, run:

```plaintext
ComposeCtl.exe configuration set --static_http_headers ""
```

To disable rest_http_headers, run:

```plaintext
ComposeCtl.exe configuration set --rest_http_headers ""
```

### 4.4 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 Compose 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:

```plaintext
ComposeCtl.exe configuration set --authentication_method sso-kerberos
```
4 Security considerations

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

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

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

4.5 Changing the master user password

All passwords are encrypted using a one-time randomly generated master key. The master key is stored automatically in the root repository of Compose (`<product_dir>/data/projects/GlobalRepo.sqlite`).

The master key is encrypted by a user key, which in turn, is derived from a master password entered by the user. By default, the Master User Password is randomly generated by Compose. The best practice, however, is to change the Master User Password, as this will allow Compose projects and configuration settings to be imported to another machine without needing to re-enter the project credentials.

It may also be convenient to use the same Master User Password within a trusted environment. In other words, if the same administrators control both the production and the testing environments, using the same Master User Password in both environments will facilitate the transfer of projects with credentials between the testing and production environments.

The user key is stored in the `muk.dat` file located in `<product_dir>/data/`.

All of the commands listed below must be run as Admin from:

```
<product_dir>/bin
```

To generate a random 32 character password:

Issue the following command:

```
ComposeCtl.exe utils genpassword
```

To change the randomly generated master user password:

1. Issue the following command:

```
ComposeCtl.exe masterukey set --password <new_master_password>
```
4 Security considerations

If you add the `--prompt` parameter to the command and omit the `--password` parameter, 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:**

```
ComposeCtl.exe masterkey set --prompt
```

2. Restart the Compose service.

To change a user-defined master user password:

1. Issue the following command:

```
ComposeCtl.exe masterkey set --current-password <current_master_password> --password <new_master_password>
```

If you add the `--prompt` parameter to the command and omit the `--password` and `--current-password` parameters, the CLI will prompt you for the required passwords. When you enter the passwords, they will be obfuscated. This is especially useful if you do not want passwords to be retained in the command prompt history.

**Syntax:**

```
ComposeCtl.exe masterkey set --prompt
```

2. Restart the Compose service.
5  Data Warehouse projects

This section explains how to set up data warehouse projects.

In this section:

- Defining a Qlik Replicate task (page 34)
- Adding and managing data warehouse projects (page 35)
- Getting started with Data Warehouse projects (page 97)
- Setting up a data warehouse connection (page 104)
- Setting up Landing Zone and Data Source connections (page 124)
- Creating and managing the model (page 141)
- Creating and managing the data warehouse (page 177)
- Creating and managing data marts (page 211)
- Creating and managing command tasks (page 238)
- Controlling and monitoring tasks and workflows (page 240)
5.1 Defining a Qlik Replicate task

In order to work with Compose, you first need to define a Qlik Replicate task that replicates the source tables from the source endpoint to a landing zone in the data warehouse (defined as the target endpoint in the Replicate task). The landing zone should then be defined as the data source for the Compose project.

For information on which endpoints can be used in a Replicate task that lands data for Compose, see Supported data warehouses (page 364).

⚠ Configuring multiple Replicate tasks with the same landing zone is not supported.

The steps below highlight the settings that are required when using Qlik Replicate with Compose. For a full description of setting up tasks in Qlik Replicate, please refer to the Qlik Replicate Help.

Prerequisites

When Oracle is defined as the source endpoint in the Replicate task, full supplemental logging should be defined for all source table columns that exist on the target and any source columns referenced in filters, data quality rules, lookups, and expressions.

Limitations and considerations

- Replicate allows you to define global transformations that are applied to source/Change tables during task runtime. The following global transformations, however, should not be defined (as they are not compatible with Compose tasks):
  - Rename Change Table
  - Rename Change Table schema
- The Create target control tables in schema option in the Replicate task settings' Control Table tab is not supported.
- Support for the JSON and XML data types is limited to the Snowflake VARIANT data type. Therefore, apart from the Snowflake VARIANT data type, columns that are usually created with these data types (by the Replicate target endpoint) should be created as STRINGs instead. Therefore, columns that are usually created with these data types (by the Replicate target endpoint) should be created as STRINGs instead. This can be done automatically within Replicate using a data type transformation. For information on which target endpoints support JSON and XML data types as well as instructions on how to create a data type transformation, please refer to the Replicate Help.
- As Compose requires a full-after image to be able to perform Change Processing, the following Replicate source endpoints are not directly supported (as they do not provide a full-after image):
  - SAP HANA (log based)
  - Salesforce
5.2 Adding and managing data warehouse projects

This section describes how to add and manage a data warehouse project.

In this section:

- **Adding data warehouse projects (page 36)**
- **Managing and monitoring projects (page 262)**
- **Project settings (page 38)**
- **Resetting projects (page 44)**
- **Project deployment (page 45)**
Adding data warehouse projects

Adding a new project is the first task you need to undertake in order to work with Qlik Compose.

There are two types of project:

- Data Warehouse - for ingesting data from multiple sources and creating analytics-ready data marts.
- Data Lake - for ingesting data from multiple sources and moving it to a storage system for analytics.

This topic guides you through the steps required to set up a data warehouse project. For instructions on setting up a Data Lake project, see Adding data lake projects (page 260).

You can set up as many projects as you need, although the ability to actually run tasks is determined by your Compose license.

Adding a Data Warehouse project

To add a new Data Warehouse project:

1. Click the New Project toolbar button. The New Project wizard opens.
2. In the Project Name tab, specify the following and then click Next:
   - Name: The project name.
   - Environment Type: Optionally, change the default environment type.
   - Environment Title: Optionally, specify an environment title.
   For information about the environment settings, see Environment tab (page 41).

   The following names are reserved system names and cannot be used as project names: CON, PRN, AUX, CLOCK$, NUL, COM1, COM2, COM3, COM4, COM5, COM6, COM7, COM8, COM9, LPT1, LPT2, LPT3, LPT4, LPT5, LPT6, LPT7, LPT8 and LPT9.

3. Select Data Warehouse as your project type and then click Finish.
4. The project panels will be displayed.
5. Add at least one source database and a data warehouse as described in Setting up Landing Zone and Data Source connections (page 124) and Setting up a data warehouse connection (page 104) respectively.

6. Create a model as described in Creating and managing the model (page 141).

7. Set up the data warehouse as described in Creating and managing the data warehouse (page 177).

8. Set up the data mart as described in Creating and managing data marts (page 211).

Managing and monitoring projects

The table below describes the available project management options.

Project management actions are performed in the main Compose window. To switch from a specific project to the main window, click the downward arrow to the right of the project name and then select All Projects from the drop-down menu.

### Project management procedures

<table>
<thead>
<tr>
<th>To</th>
<th>Do this</th>
</tr>
</thead>
<tbody>
<tr>
<td>Edit a project</td>
<td>Any of the following:</td>
</tr>
<tr>
<td></td>
<td>• Double-click the project.</td>
</tr>
<tr>
<td></td>
<td>• Right-click the project and select Designer.</td>
</tr>
<tr>
<td></td>
<td>• Select the project and then click the Open toolbar button.</td>
</tr>
<tr>
<td>Monitor a project</td>
<td>Any of the following:</td>
</tr>
<tr>
<td></td>
<td>• Right-click the project and select Monitor.</td>
</tr>
<tr>
<td></td>
<td>• Double-click the project and select the Monitor tab on the right of the console.</td>
</tr>
</tbody>
</table>
5  Data Warehouse projects

<table>
<thead>
<tr>
<th>To</th>
<th>Do this</th>
</tr>
</thead>
<tbody>
<tr>
<td>Create a deployment</td>
<td>Any of the following:</td>
</tr>
<tr>
<td>package</td>
<td>- Right-click the project and select <strong>Create Deployment Package</strong>.</td>
</tr>
<tr>
<td></td>
<td>- Select the project and then select <strong>Create Deployment Package</strong> from the Deployment toolbar.</td>
</tr>
<tr>
<td></td>
<td><strong>See also:</strong> Project deployment (page 45) (Data Warehouse projects) and Project deployment (page 269) (Data Lake projects).</td>
</tr>
<tr>
<td>Delete a project</td>
<td>Any of the following:</td>
</tr>
<tr>
<td></td>
<td>- Right-click the project and select <strong>Delete</strong>.</td>
</tr>
<tr>
<td></td>
<td>- Select the project and then click the <strong>Delete</strong> toolbar button.</td>
</tr>
<tr>
<td>View or change user</td>
<td>Right-click the project and select <strong>User Permissions</strong>.</td>
</tr>
<tr>
<td>permissions</td>
<td><strong>Relevant for Data Warehouse projects only.</strong></td>
</tr>
<tr>
<td></td>
<td><strong>See also:</strong> User permissions (page 348).</td>
</tr>
</tbody>
</table>

Project settings

You can change the project settings according to your needs.

**To access the project settings:**

1. Open your project as described in Managing and monitoring projects (page 262).
2. Click the downward arrow to the right of the project name and select **Settings** from the drop-down menu.

   The **Settings** window opens, displaying the following tabs:
   - **General tab (page 38)**
   - **Naming tab (page 40)**
   - **Environment tab (page 41)**
   - **Table creation modifiers tab (page 43)**

General tab

In this tab, the following settings are available:

Miscellaneous

- **Generate DDL scripts but do not run them**: By default, Compose executes the CREATE, ADJUST and DROP statements immediately upon user request. When you select this option, Compose will only generate the scripts but not execute them. This allows you to review and edit the scripts before they are executed.
  For example, if you want your data warehouse/storage tables to contain partitions, you will need to edit the CREATE statement to create the partitions.
You can view, copy and download the DDL scripts as described in Viewing and downloading DDL scripts (page 94).

When this option is selected, you need to do the following to see the results:

- After running the scripts, clear the metadata cache as described in Clearing the data warehouse metadata cache (page 210).
- When this option is selected, you need to press [F5] (i.e. refresh the page) in order for the web console to display the updated list of tables. This can be done either before running the scripts (recommended) or after running the scripts. Note that until you refresh the browser, the information in the web console will only be partially updated.

- **Ignore Mapping Data Type Validation**: By default, Compose issues a validation error when a landing table is mapped to a logical entity with a different data type. You can select this option to allow the mapping of different data types. Note that you should only select this option if you need to map landing table data types to compatible (though not identical) logical entity data types.

- **Write metadata to the TDWM tables in the data warehouse**: When this option is selected (the default unless Amazon Redshift is the data warehouse), Compose writes the metadata for the data warehouse tables to the following tables:
  `<schema>.TDWM_Tables` and `<schema>.TDWM_COLUMNS`.

  Centralizing the metadata in two dedicated tables makes it easier for external metadata tools to analyze the metadata. The metadata is also written to the local Compose repository, so clearing this option (if performance issues are encountered) will not affect Compose functionality in any way.

- **Do not display the default workflows in the monitor**: Select this option if you want to prevent the default workflows from being executed.

**Dates**

- **Lowest Date**: The value stored in the "From Date" column. This is the date when the version started.

- **Highest Date**: The value stored in the "To Date" column. This is the date when the version ended.

**Current Time Convention**

When a source record's timestamp cannot be determined, select one of the following to use instead:

- Current time in UTC (the default for new projects)

To preserve backward compatibility when upgrading or deploying old projects, local server time is the default. Before changing this option, make sure that existing data will not be impacted.

- Current time in server local time

For existing objects, Compose will not be able to determine a source record's timestamp if both of the following are true:
The "From Date" columns are not mapped
The task is set up to perform Incremental Load

Naming tab
In this tab, you can change the default "From Date" and "To Date" column names, as well as the prefixes and suffixes used to identify tables, views, and columns.

If you change the prefix or suffix of existing tables (e.g. data warehouse tables), you need to drop and create the data warehouse and data mart tables.

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Suffix for Replicate Change Tables</td>
<td>The suffix used to identify Replicate Change Tables in the landing zone of the data warehouse.</td>
</tr>
<tr>
<td>Prefix for data warehouse tables</td>
<td>The prefix used to identify tables in the Data Warehouse.</td>
</tr>
<tr>
<td>Prefix for data warehouse views</td>
<td>The prefix used to identify views in the Data Warehouse.</td>
</tr>
<tr>
<td>Suffix for archived Replicate Change Tables</td>
<td>The suffix used to identify archived Change Tables in the specified database.</td>
</tr>
<tr>
<td></td>
<td>For more information on archiving Change Tables, see <a href="#">After applying changes</a>.</td>
</tr>
<tr>
<td>Prefix for data mart tables</td>
<td>The prefix used to identify tables in the data mart.</td>
</tr>
<tr>
<td>Suffix for exception mart tables</td>
<td>The suffix used to identify error tables in the data warehouse. These tables contain data that was rejected by a data quality rule.</td>
</tr>
<tr>
<td>Suffix for hub tables</td>
<td>The suffix used to identify hub tables in the Data Warehouse. Hub tables contain History Type 1 columns. History Type 1 column do not contain any version history as opposed to History Type 2 columns that do.</td>
</tr>
<tr>
<td>Suffix for satellite tables</td>
<td>The suffix used to identify satellite tables in the Data Warehouse. Satellite tables contain History Type 2 columns. History Type 2 columns keep a history of the data version by adding a new row whenever the data is updated.</td>
</tr>
<tr>
<td>&quot;From Date&quot; column name</td>
<td>The name of the &quot;From Date&quot; column. This column is added to tables that contain attributes (columns) with a History Type 2. The column is used to delimit the range of dates for a given record version. This name cannot be used in other columns.</td>
</tr>
</tbody>
</table>

Setup and User Guide - Qlik Compose, August 2021
### Environment tab

In this tab, you can:

- Specify information about your environment, part of which will be displayed as a banner at the top of the window when you open the project.
- Determine the number of database connection to open concurrently.

After providing the following information, click **OK** to save your settings:

- **Environment type**: Select one of the following types according to your environment type: *Development, Test, Acceptance, Production, Other*. This information will not be displayed in the banner.
- **Environment title**: Specify a title for your environment. The title will be displayed in the banner at the top of the console.
- **Project title**: Specify a title for your project. The project title will be shown in the console banner. If both an **Environment Title** and a **Project Title** are defined, the project title will be displayed to the right of the environment title.

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;To Date&quot; column name</td>
<td>The name of the &quot;To Date&quot; column. This column is added to tables that contain attributes (columns) with a History Type 2. The column is used to delimit the range of dates for a given record version. This name cannot be used in other columns.</td>
</tr>
</tbody>
</table>

- **The Project title option requires Compose August 2021 Patch Release 12 or later.**
- **When a project is deployed to a new environment, the environment title and environment type in the new environment will not be overridden.**

The following image shows the banner with both an **Environment title** and a **Project title**:

![Banner with Environment and Project title](image)

The banner text is shown without the **Environment title** and **Project title** labels. This provides greater flexibility as it allows you add any banner text you like, regardless of the actual label name. For example, specifying **Project owner: Mike Smith** in the **Project title** field, will display that text in the banner.
Creating or Dropping Data Warehouse Tables

Limit the number of database connections to: The higher the number of database connections, the more data warehouse tables Compose will be able to create or drop in parallel. While increasing the default should improve performance, it might also impact other database applications. It is therefore not recommended to increase the default unless you encounter performance issues.

The environment properties can be exported and imported to a new project, but cannot be imported to an existing project.

Task recovery

You can set SQL state classes and error codes, on the occurrence of which, a task will be retried.

You can set the following parameters:

- **Maximum retry count**: The number of times to retry a task before exiting with failure. Increasing the number of retries will impact system resources. Therefore, only increase the default value if you expect tasks to recover after the default number of retries.
- **Interval between retry attempts (sec)**: The time to wait between retry attempts. Increasing the interval will consume more system resources. Therefore, only increase the default value if it is critical that the task recover as soon as possible.
- **Retry on these SQL state classes**: The default is 08 (connection exceptions). You can add additional classes as desired. Classes should be separated with a comma.

  **Example**: 08,22,2F

- **Retry also on these error codes**: The default is 1205 (which occurs when a table is locked by another process). You can add additional error codes as desired. Error codes should be separated with a comma.

  **Example**: 1205,2020,233

Limitations and considerations:

- ODBC statements comprise a small part of the task execution sequence. However, as the task retry mechanism is JDBC-based, ODBC statements will not be retried even if the specified SQL state/error code is encountered.

Task and Workflow Information Retention

You can set the maximum number of runs to keep task and workflow logs and messages. The default is 100. Task information includes logs, the number of inserted/updated rows per table, errors, and various other runtime messages. If you find that the number of accumulated logs and messages is degrading performance, reducing this value might help.

This setting requires Compose August 2021 SR1 or later.
Table creation modifiers tab

By default, Compose creates tables in the data warehouse using the standard CREATE TABLE statement. However, organizations often need tables to be created with custom properties for better performance, special permissions, custom collation, and so on. For example, in Microsoft Azure Synapse Analytics, it is possible to create a table as a HEAP, which is optimized for smaller tables. By default, Compose creates tables in Microsoft Azure Synapse Analytics as a CLUSTERED COLUMNSTORE INDEX, which offers the best overall query performance for large tables.

In the Table creation modifiers tab, you can append table creation modifiers as SQL parts to the CREATE TABLE statement. You can set table creation modifiers for both data warehouse tables and for data mart tables. In the data warehouse, separate modifiers can be set for Hub and Satellite tables while in the data mart, separate modifiers can be set for fact and dimension tables. Once set, all tables will be created using the specified modifiers, unless overridden at the entity level.

To set table creation modifiers:

1. Select the Custom option for any of the available table types.
2. Click the Edit button to open the Table Creation Modifier editor.
3. Enter the SQL parts you want to append to the CREATE TABLE statement.
4. Optionally, but strongly recommended, validate the SQL in an external validation tool that supports your specific database and version.

Compose does not provide any way of validating your SQL. Therefore, make sure to validate the SQL before deploying in a production environment.

5. Click OK to close the editor and save your SQL.

Limitations

If you change an existing table creation modifier, you will not be prompted to adjust affected tables when validating the model. If you want to apply the change to existing tables, dropping and recreating all tables might not be an issue in a development environment. However, in a production environment (where dropping and recreating all tables might not be a viable option), you will need to adjust the tables outside of Compose.

Example of a Valid Table Creation Modifier

In the following example, the Compose CREATE TABLE statement (rows 1-5) is appended with an SQL part instructing Compose to create the table as a HEAP (row 6).

```sql
CREATE TABLE MyTable
{
  column1 integer,
  column2 varchar(50),
}
WITH (HEAP)
```
For an explanation of how to define table creation modifiers for individual data warehouse tables, see *Defining Table Creation Modifiers* (page 166).

For an explanation of how to define table creation modifiers for individual fact tables, see *Example of a Valid Table Creation Modifier* (page 225).

For an explanation of how to define table creation modifiers for individual dimension tables, see *Example of a Valid Table Creation Modifier* (page 230).

**Resetting projects**

You can reset projects as required. This can be useful in the project development stage as it allows you to easily delete unwanted project elements.

*Be careful not to reset a project and delete data in a production environment.*

**To reset a project:**

1. Open your project as described in *Managing and monitoring projects* (page 262).
2. Click the downward arrow to the right of the project name and select **Reset Project** from the drop-down menu.
   The **Reset Project** window opens.
3. Select which elements to reset according to your project type.
   - Model (Entities, Relationships, Attribute Domains), mappings, and data mart definitions
     For more information on models, see *Creating and managing the model* (page 141).
   - Reusable transformations
     For information on the reusable transformations, see *Defining reusable transformations* (page 176).
   - Global mappings
     For more information on global mapping, see *Managing global mappings* (page 147).
   - Data warehouse and data mart tables
     For more information on data warehouses and data marts, see *Creating and managing the data warehouse* (page 177) and *Creating and managing the data warehouse* (page 177) respectively.
   - Command tasks
     For more information on data warehouses and data marts, see *Creating and managing command tasks* (page 238).
   - DDL Scripts
     For more information on DDL scripts, see *Project settings* (page 38) and *Viewing and downloading DDL scripts* (page 94).
   - Drop Archive Tables
     For more information on Archive Tables, see *Defining landing zones* (page 132).
4. Click **Reset Project** and then click **Yes** when prompted to confirm your request.
Project deployment

Project deployment packages can be used to back up projects or migrate projects between different environments (e.g. testing to production). As a deployment package is intended to be deployed in a new environment, it contains the Data Warehouse and data source definitions, but without any passwords. The deployment package also does not contain any data from the Data Warehouse or data mart, only the metadata. The deployment package also contains the project metadata and mapping information, which should be consistent with the landing zone tables in the new environment.

For a complete list of objects contained in the deployment package, see Exporting a project (page 76).

Creating deployment packages

This section explains how to create a project deployment package.

To create a deployment package

1. Choose one of the following methods:
   - In the main Compose window, right-click the desired project and select **Create Deployment Package** from the context menu.
   - In the main Compose window, select the desired project. Then, click the Deployment toolbar button and select **Create Deployment Package** from the drop-down menu.
   - In the project window, select **Deployment > Create Deployment Package** from the project drop-down menu.

   The **Create Deployment Package - <Project_Name>** window opens.

2. Provide a **Version** number and a **Description** in the designated fields and then click **OK**.

A ZIP file containing a JSON file (i.e. the project settings) and a **readme.txt** file will be saved to your browser's default download location. The ZIP file name is in the following format: **<Project_Name>_deployment_<Date>___<Time>.zip**

The **readme.txt** file contains the following information about the deployment package: project name, export date, exporter user name, deployment version, and description.

Deploying packages

This section explains how to deploy a project deployment package. This section explains how to deploy a project deployment package. You can only deploy packages to an existing project. Therefore, before deploying a project, create a new project with the user name and password required for connecting to the Data Warehouse database and the Landing Zone database (if defined) in the new environment. In addition, the Landing Zone databases in the target project must have the same display name (defined in the Compose console) as the corresponding databases in the source project. Note that as database settings are usually environment specific, the database settings in the target project will not be overwritten by those of the source project.

**Deploying a project between different database types is not supported. For example, you cannot create a package in SQL server and deploy it to an Oracle database.**
When deploying, Compose does not override existing connection parameters. This enables you to easily migrate projects from test to production, for example, without needing to change user names, passwords or IP addresses.

If preferred, you can create an empty project and provide the required credentials after the deployment completes. In this case, an error message prompting you for the missing credentials will be displayed after the deployment completes.

To deploy a project:

1. Copy the ZIP file created in Creating deployment packages (page 45) to a location that is accessible from the Compose machine.
2. Open Compose and choose one of the following methods:
   - In the main Compose window, select the desired project. Then, click the Deployment toolbar button and select Deploy from the drop-down menu.
   - In the project window, select Deployment > Deploy from the project drop-down menu.
   The Deploy window opens.
3. Either drag and drop the file on the window.
   OR  
   Click Select and browse to the location of the deployment package. In the Open window, either double-click the deployment package ZIP file or select the file and click OK. The package details will be displayed.
4. Click Deploy to deploy the package. When prompted to replace the existing project, confirm the operation. The project will be deployed.

When deploying a project defined with multiple Replicate Servers to any of the following:

- A project without any Landing Zone databases
- A project which is missing one or more Landing Zone databases defined in the source project

Then the Landing Zone settings from the source project will be used, but the missing databases will be created without a password and Replicate Server. These will need to be configured manually.

Migrating objects as CSV files

Relevant to Data Warehouse projects only.

Migrating Compose objects as CSV files, provides a level of granularity that is not available when using the standard project export and deployment options. Instead of importing the entire project, you can import specific objects and then apply periodic updates as needed.

The following objects can be migrated:
5 Data Warehouse projects

- Model objects: Entities, Attributes, Attributes Domain, Relationships, Reusable transformations, and Reusable transformation parameters
- Mappings and Mapping Metadata
- Data Mart objects: Fact tables, Fact attributes, Star schemas, Dimensions, and Dimension attributes

In this section:

- Overview (page 47)
- Migrating models (page 49)
- Migrating mappings (page 54)
- Migrating tasks (page 57)
- Migrating data marts (page 62)
- Migrating reusable transformations (page 68)
- Commands for exporting/importing objects (page 69)
- Commands for comparing and applying objects (page 72)

Overview

The ability to export object definitions to a CSV file and then import them to another environment provides many benefits, enabling:

- Migration of data from a custom database table and/or Excel sheets to Compose
- Data to be reviewed by business analysts who are not able to (or do not want to) access Compose
- Synching with third-party tools that output data in CSV format
- Comparison of versions in order to review changes
- Resolving of object-specific issues in a development environment and then deploying to the production environment, even when they are not completely in sync
- Sharing of resources between projects in the same environment
- Granular version management of specific objects such as mappings

Manually typing these definitions into Compose would be a laborious and time-consuming undertaking; using the CLI however, this can be done in a matter of minutes. Following the initial import, customers who need to apply selective updates to the target environment (such as adding attributes with their descriptions), can do so using the Compare and Apply CLI commands.

A typical workflow would be as follows:

1. Run the export command to output the source object(s) (to CSV files).
2. Run the import command to bring the objects into the target environment.
3. Following changes to the source environment, run the export command to output the source object (s) to (CSV files).
4. Run the Compare command to see the differences between the exported objects and the corresponding target objects.
Alternatively, run the Compare command on the source environment to see the differences between previous and current source project versions; then determine which changes need to be migrated to the target project. This approach is useful if changes were made directly to the target project as it allows you to retain the custom changes while still applying changes to other objects.

5. Review the changes and make any edits as necessary.
6. Run the Apply command to apply the changes to the target environment.
7. Periodically repeat steps 3-6 as necessary.

An understanding of the CSV file structure and their impact on the target environment is crucial, not only for customers who wish to create these files manually, but also for ensuring the import/apply operations succeed with the expected results. For this reason, this section first discusses the CSV file structure of the supported objects and only then provides instructions for performing the actual CLI operations.

- Only import of new objects is supported.
- UPDATEs and DELETEs will be ignored by the Apply command (which currently only supports ADD operations), but can still be applied manually if needed.

Valid CSV file formats
CSV files must be in a valid format; otherwise, the import/compare/apply operation will fail.

CSV formatting rules:
- Fields must be separated with a comma.
- The first line must be a header line which contains the column names.
- The escaping convention is similar to Excel:
- If a field contains a comma, then the comma must be wrapped with quotation marks, for example: "a,b"
- If the value contains a quotation mark, then the quotation mark must be doubled. For example: "Mike ""The Hammer"" Smith"
- If the value contains a new line, then it must be wrapped with double quotes. For example:
  "Field owned by Mike."
  "Comment added by Shelley."

Stored objects
Several objects may have multi-lines, commas, and other such complexities.

Expressions, for example, are stored as two fields:

First field - expression value:

Example:

${x1}*${x2}

Note: The expression may need to be escaped.
Second field - semicolon parameters mappings:

Example:

x1:unit price;x2:quantity

Migrating models

Migrating a Compose Model allows you to:

- Import existing column definitions (i.e. definitions stored independently of Compose) to a Compose project
- Reuse the same Model across several projects or Compose installations

Model objects

A Compose Model consists of the following objects:

- Entities (entities.csv)
- Attributes (attributes.csv)
- Attributes Domain (attributesDomain.csv)
- Relationships (relationships.csv)

During the export process, each of these objects is exported to a separate CSV file.

You can either import the Model in its entirety or only specific elements, according to your needs. You can also manually create a CSV file containing a Model element (or edit an existing file) and then import it to a Compose project.

Guidelines for exporting a Compose model

Note the following:

- For Boolean fields, accepted values are True/False
- Data type is the Compose logical type
- The order of writing the attributes is according to the ordinal in the entity. Primary Keys will be shown first, even if they were not first in the source model
5 Data Warehouse projects

- Attribute domain and entity order is alphabetical in the CSV
- Relationship order is by entity name (alphabetical) and according to their order in the entity

Guidelines for importing a Compose model

Note the following:

- For boolean fields, accepted values are True/False
- Column order has no meaning; only column names, which are case insensitive, unless defined differently
- If the entities.csv file is missing, the entities can be inferred from the attributes.csv file (with no description). In such cases, duplicate objects are verified to be the same and added only once. For example, if there are several rows with entity name myEntity, the entity will only be added once
- Data type is the Compose logical type
- Relationship details override the underlying attributes information (e.g. history type, is key, etc.)
- When importing the attributes.csv file, a new entity will not be created if wasn’t already created in the entities.csv file, and its attributes all have relationships to other entities. In such cases, you should create the entity manually or add it to the entities.csv file
- When importing the relationships.csv file, existing attributes will be replaced if:
  child attribute prefix+attribute name = relationship_prefix+parent_attribute_name_prefix+parent_attribute_name
  For example, the attribute named CustomerDesignatedID will be replaced by the relationship where:
  - ID is the name of the attribute in the parent entity and
  - Customer is the prefix of the relationship and
  - Designated is the prefix of the parent attribute.
  Note that attributes marked as relationships will be skipped when imported from attributes.csv as they must derive their data type from the Attributes Domain.

Entities CSV file format

<table>
<thead>
<tr>
<th>Column Name</th>
<th>Required</th>
<th>If column/value is missing</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Entity Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Entity Description</td>
<td>No</td>
<td>Empty</td>
<td>-</td>
</tr>
</tbody>
</table>

Attributes CSV file format

If Attribute Domain is missing in the attributes.csv file, there must be a data type. Attribute domains that differ but have the same name will be appended with the suffix _01.

<table>
<thead>
<tr>
<th>Column Name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If value is missing</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Entity Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
</tbody>
</table>
## 5 Data Warehouse projects

<table>
<thead>
<tr>
<th>Column Name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If value is missing</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Attribute Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Description</td>
<td>No</td>
<td>Empty</td>
<td>Accept</td>
<td></td>
</tr>
<tr>
<td>Is Key</td>
<td>No</td>
<td>No attributes will be defined as keys. In such a case, validation will fail as at least one key needs to be defined.</td>
<td>No Key</td>
<td>On import, Compose does not validate that each entity has at least one key attribute (required). If you import entities without key attributes, then you must define a key attribute in Compose after importing the entity. Otherwise, Model validation will fail.</td>
</tr>
<tr>
<td>Attribute Domain</td>
<td>No</td>
<td>If it is missing, Data Type must exist and attribute domains will be built or used the same as they are during discovery.</td>
<td>Reject</td>
<td>The Attribute Domain name or the word &quot;Relationship&quot;</td>
</tr>
<tr>
<td>Data Type</td>
<td>No</td>
<td>If it is missing, the Attribute Domain must exist. An error will be returned if the data type does not match the attribute domain or if both exist.</td>
<td>Reject</td>
<td>Use combined syntax: Varchar(50) Decimal(10,2)</td>
</tr>
<tr>
<td>Prefix</td>
<td>No</td>
<td>Empty</td>
<td>Accept (an empty field means there will be no prefix)</td>
<td>-</td>
</tr>
<tr>
<td>History Type</td>
<td>No</td>
<td>Key - Type 1, Not key - type 2</td>
<td>Reject</td>
<td>Values are Type 1 or Type 2. Yes (Type 2) and No (Type 1) are also allowed.</td>
</tr>
<tr>
<td>Satellite/Hub</td>
<td>No</td>
<td>Key is Hub; other attributes are SAT 1</td>
<td>Reject</td>
<td>Hub/1/2/3</td>
</tr>
</tbody>
</table>
### Column Name

<table>
<thead>
<tr>
<th>Column Name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If value is missing</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Expression</td>
<td>No</td>
<td>No expressions in any attribute</td>
<td>No expression in any attribute</td>
<td>-</td>
</tr>
<tr>
<td>Expression Params</td>
<td>No</td>
<td>All attribute-parameter mappings are trivial (same name)</td>
<td>All attribute-parameter mappings are trivial (same name)</td>
<td>-</td>
</tr>
<tr>
<td>Add after</td>
<td>No</td>
<td>Not relevant.</td>
<td>Not relevant.</td>
<td>Instructs Compose to add the attribute after a specific attribute. The column is only relevant when adding attributes. When the column is empty, the corresponding attributes should be added at the end (according to their order in the CSV file). Note that it may refer to any attribute that is defined above the current row. For example, assuming a record contains the attributes address, height, and weight, specifying the following: school, add after=weight ID, add after=school email, add after=empty Will result in the following order: address, height, weight, school, ID, email</td>
</tr>
</tbody>
</table>

### Attributes Domain CSV file format

<table>
<thead>
<tr>
<th>Column Name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If value is missing</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
</tbody>
</table>
# Data Warehouse projects

<table>
<thead>
<tr>
<th>Column Name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If value is missing</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Description</td>
<td>No</td>
<td>Empty</td>
<td>The specific attribute value may be empty as well.</td>
<td>-</td>
</tr>
<tr>
<td>Data Type</td>
<td>No</td>
<td>Reject</td>
<td>Reject</td>
<td>Use combined syntax: Varchar(50) Decimal(10,2)</td>
</tr>
</tbody>
</table>

## Relationships

### Relationships CSV mapping rules

<table>
<thead>
<tr>
<th>Column Name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If value is missing</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Child Entity</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>This is the more detailed entity (e.g. OrderDetails). Case insensitive.</td>
</tr>
<tr>
<td>Parent Entity</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>This is the less detailed entity (e.g. Products). Case insensitive.</td>
</tr>
<tr>
<td>Prefix</td>
<td>No</td>
<td>Empty</td>
<td>If all the originating attributes have the same prefix, use it here. A relationship may not have a prefix to its underlying attributes. The specific attribute value may be empty as well.</td>
<td>-</td>
</tr>
</tbody>
</table>
5  Data Warehouse projects

<table>
<thead>
<tr>
<th>Column Name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If value is missing</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Position After</td>
<td>No</td>
<td>Add the relationships to the end. Relationships are ordered according to their order in the file.</td>
<td>Same as missing column</td>
<td>The attribute name or relationship to position it after or &quot;0&quot; to position it first.</td>
</tr>
<tr>
<td>Description</td>
<td>No</td>
<td>Empty</td>
<td>Specific attribute may have empty value as well</td>
<td></td>
</tr>
<tr>
<td>Is Key</td>
<td>No</td>
<td>No attributes will be defined as keys. In such a case, validation will fail as at least one key needs to be defined.</td>
<td>No key</td>
<td></td>
</tr>
<tr>
<td>History Type</td>
<td>No</td>
<td>Key: Type 1</td>
<td>Reject</td>
<td>Values are Type 1 or Type 2.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Non-key: Type 2</td>
<td></td>
<td>Yes (Type 2) and No (Type 1) are also allowed.</td>
</tr>
<tr>
<td>Satellite/Hub</td>
<td>No</td>
<td>Key: Hub</td>
<td>Reject</td>
<td>Hub/1/2/3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Other attributes: SAT 1</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Migrating mappings

Migrating mappings allows you to:

- Export mapping metadata and mappings from a Compose project to CSV files. Mapping metadata will be exported to `mappingsMetadata.csv` while mappings will be exported to `mappings.csv`. The former shows the table mappings while the latter show the column mappings.
- Import new mappings that do not exist in the current Compose project.
- Reuse the same mappings across several projects or Compose installations.

Mappings export guidelines

When exporting mappings and mapping metadata from Compose, it is important to note the following:

- The export of mappings is allowed for users with the Viewer security role.
- The order of writing the mapping metadata is according to metadata name alphabetically (e.g. `map_orders` appears after `Map_Customers`).
- The order of writing a mapping is according to target columns (same as Model ordinal).
- Source columns which are not mapped to anything will not appear in the exported file.
- All target columns will appear in the mappings even if they were not mapped.
Mappings import guidelines

When importing mappings and metadata to the Model, it is important to note the following:

- If required, you can import one CSV file at a time: mappingsMetadata.csv or mappings.csv.
- Column order has no meaning; only column names (case insensitive).
- When importing metadata, Compose validates that the targets exist in the Model. Source columns are not validated on import.
- If the source schema, table, view or query don't exist, they will be validated after the import.
- Importing mappings will fail in the following scenarios:
  - If the target entity doesn't exist in the Model.
  - If the compose database object doesn't exist.
  - If the mapping attribute doesn't exist in the Model.

Valid CSV file formats

The CSV files must be in a valid format. For more detailed information, see the notes in Valid CSV file formats (page 48).

Mapping metadata CSV file format

<table>
<thead>
<tr>
<th>Column Name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If value is missing</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Landing Zone</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>The Compose &quot;source database&quot; name (excluding the word &quot;landing&quot;)</td>
</tr>
<tr>
<td>Database</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Schema</td>
<td>No</td>
<td>If there is no</td>
<td>If no schema</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Schema column,</td>
<td>is specified,</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>either the default</td>
<td>either the default</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>schema will be used</td>
<td>schema will be used</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>or none (as some</td>
<td>or none (as some</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>databases do not</td>
<td>databases do not</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>have separate</td>
<td>have separate</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>schemas)</td>
<td>schemas)</td>
<td></td>
</tr>
<tr>
<td>Source Type</td>
<td>No</td>
<td>Table</td>
<td>Reject</td>
<td>Table/View/Query</td>
</tr>
<tr>
<td>Column Name</td>
<td>Required</td>
<td>If column is missing</td>
<td>If value is missing</td>
<td>Comments</td>
</tr>
<tr>
<td>------------------</td>
<td>----------</td>
<td>----------------------</td>
<td>--------------------</td>
<td>-----------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Source Object</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>If Source Type is <strong>Table</strong> or <strong>View</strong>, the Source Object is its name. If Source Type is <strong>Query</strong>, the rules and limitations described in <em>Valid CSV file formats</em> (page 48) will be applied.</td>
</tr>
<tr>
<td>Target Entity</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>-</td>
</tr>
<tr>
<td>Filter</td>
<td>No</td>
<td>No filter</td>
<td>No filter</td>
<td>Example of Semicolon parameters mappings: x1:unit &gt; price; x2:quantity</td>
</tr>
<tr>
<td>Filter Params</td>
<td>No</td>
<td>Same name for attribute-parameter, like expression below</td>
<td>Same name for attribute-parameter, like expression below</td>
<td></td>
</tr>
<tr>
<td>Mapping Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Target Column</td>
<td>Yes</td>
<td>Reject</td>
<td>No mapping if there is also no expression or lookup.</td>
<td></td>
</tr>
<tr>
<td>Mapping Type</td>
<td>No</td>
<td>Field mapping</td>
<td>Allow the field to be empty if there is no field mapping, no expression, and no lookup.</td>
<td></td>
</tr>
<tr>
<td>Field Mapping</td>
<td>No</td>
<td>Reject</td>
<td>No mapping to this attribute.</td>
<td></td>
</tr>
<tr>
<td>Expression</td>
<td>No</td>
<td>No expression</td>
<td>No expression in this attribute.</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Example of an expression value: ${x1}*$${x2}$</td>
</tr>
</tbody>
</table>
## Migrating tasks

You can migrate data warehouse tasks, data mart tasks, and custom ETLs (tasks) from one environment to another, while preserving custom objects in the target environment. This is especially useful for customers who wish to incrementally updated production environments with new versions from the test environment.

The following files will be exported:

For the data warehouse:

### Table: Data Warehouse projects

<table>
<thead>
<tr>
<th>Column Name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If value is missing</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Expression Params</td>
<td>No</td>
<td>All attribute-parameter mapping is trivial (same name)</td>
<td>No expression in that attribute</td>
<td>Example of Semicolon parameters mappings: x1:unit &gt; price; x2:quantity</td>
</tr>
<tr>
<td>Lookup Landing Database</td>
<td>No</td>
<td>No lookups</td>
<td>No expression in that attribute</td>
<td>Lookup Landing Database name</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Example:</strong> Northwind on MySQL</td>
</tr>
<tr>
<td>Lookup Table</td>
<td>No</td>
<td>No lookups</td>
<td>No expression in that attribute</td>
<td>Lookup table or view name in schema table format.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Example:</strong> Schemal. Orders</td>
</tr>
<tr>
<td>Lookup Type</td>
<td>No</td>
<td>No lookups</td>
<td>No expression in that attribute</td>
<td>Table or View</td>
</tr>
<tr>
<td>Lookup Condition Value</td>
<td>No</td>
<td>No lookups</td>
<td>No expression in that attribute</td>
<td>-</td>
</tr>
<tr>
<td>Lookup Condition Params</td>
<td>No</td>
<td>No lookups</td>
<td>No expression in that attribute</td>
<td>Include Lookup/Landing.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>Example:</strong> x:$Lookup$.a; y:$Landing$.CustomerID</td>
</tr>
<tr>
<td>Lookup Result Value</td>
<td>No</td>
<td>No lookups</td>
<td>No expression in that attribute</td>
<td>-</td>
</tr>
<tr>
<td>Lookup Result Params</td>
<td>No</td>
<td>No lookups</td>
<td>No expression in that attribute</td>
<td>Same format as the lookup condition parameters.</td>
</tr>
</tbody>
</table>
5  Data Warehouse projects

- `<specified export folder>/customEtl.csv` - Contains details of any (enabled or disabled) custom ETLs defined for the task.
- `<specified export folder>/taskCustomEtl.csv` - Lists any enabled custom ETLs defined for the data warehouse task.
- `<specified export folder>/taskSettings.csv` - Contains details of the task settings defined for each of the data warehouse tasks.
- `<specified export folder>/tasks.csv` - Lists the data warehouse tasks.
- `<specified export folder>/taskDataWarehouseTables.csv` - Lists the data warehouse tables and properties.
- `<specified export folder>/taskMappings.csv` - Lists the mappings used in the task.
- `<specified export folder>/SQL/DW_Custom_ETL_<custom ETL name>.SQL` - One SQL file for each custom ETL.

For each data mart:

- `<specified export folder>/<data mart name>/customEtl.csv` - Contains details of any custom pre-loading or post-loading ETLs defined for the data mart task
- `<specified export folder>/<data mart name>/taskSettings.csv` - Contains details of the task settings
- `<specified export folder>/<data mart name>/SQL/DM_Custom_ETL_<custom ETL name>.SQL` - One SQL file for each custom ETL

Considerations

Export considerations

- Parameters will be written to CSV files in alphabetical order (as they appear in the web console).

Import considerations

- Importing tasks or custom ETLs will override any existing objects with the same names.
- Importing logical entities or mappings that do not exist in the target project will result in failure.

Valid CSV file formats

The CSV files must be in a valid format. For more detailed information, see the notes in Valid CSV file formats (page 48).

Data warehouse CSV file formats

Tasks file

The tasks.csv file consists of one row per task.

<table>
<thead>
<tr>
<th>Header name</th>
<th>Mandatory</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Task name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Description</td>
<td>No</td>
<td>Empty</td>
<td>Empty</td>
<td>-</td>
</tr>
</tbody>
</table>
## 5 Data Warehouse projects

<table>
<thead>
<tr>
<th>Header name</th>
<th>Mandatory</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>No</td>
<td>Full Load Only</td>
<td>Reject</td>
<td>Full Load Only or Change Tables Only</td>
</tr>
</tbody>
</table>

### Task settings file

The `taskSettings.csv` file consists of one row per setting.

<table>
<thead>
<tr>
<th>Header name</th>
<th>Mandatory</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Task Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Setting Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Setting Value</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>-</td>
</tr>
</tbody>
</table>

### Task entities file

The `taskDataWarehouseTables.csv` file consists of one row per entity.

<table>
<thead>
<tr>
<th>Header name</th>
<th>Mandatory</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Task Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Entity Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Handle duplicates</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Boolean</td>
</tr>
</tbody>
</table>

### Task mappings file

The `taskMappings.csv` consists of one row per task for each mapping that is used in the task.

<table>
<thead>
<tr>
<th>Header name</th>
<th>Mandatory</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Task Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
</tbody>
</table>
### Custom ETL file

The `customEtl.csv` consists of one row for each custom ETL defined for the task (Pre-Loading, Multi-Table, Single Table, or Post-Loading), regardless of whether or not the ETL is enabled.

<table>
<thead>
<tr>
<th>Header name</th>
<th>Mandatory</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Description</td>
<td>No</td>
<td>Empty</td>
<td>Empty</td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Pre Loading ETL, Multi Table ETL, Single Table ETL or Post Loading ETL</td>
</tr>
<tr>
<td>Entity</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Relevant only for single table ETL</td>
</tr>
<tr>
<td>Sequence Number</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Valid values are positive integer numbers</td>
</tr>
<tr>
<td>Execute as Stored Procedure</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Boolean</td>
</tr>
</tbody>
</table>

- For each custom ETL, Compose will export/import an SQL file to: `<specified export folder>/SQL`
- The file name will be `DW_Custom_ETL_<custom ETL name>.SQL`
- If you wish to edit the file name, make sure that it only contains the following characters: A-Z, 0-9, underscore (_), or space. On import, any other character will be replaced with an underscore.

### Task custom ETL file

The `taskCustomEtl.csv` file consists of one row for each enabled custom ETL used in the data warehouse task.
5 Data Warehouse projects

Data warehouse task custom ETL

<table>
<thead>
<tr>
<th>Header name</th>
<th>Mandatory</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>DWH Task Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Custom ETL Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
</tbody>
</table>

Data mart CSV file formats

Task settings file

The taskSettings.csv file consists of one row per setting.

<table>
<thead>
<tr>
<th>Header name</th>
<th>Mandatory</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Setting Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Setting Value</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>-</td>
</tr>
</tbody>
</table>

In versions earlier than Compose August 2021 SR1, the taskSettings.csv file contained a Task Name column. Starting from Compose August 2021 SR1, the taskSettings.csv file will not include the Task Name column as the task name will be inferred from its containing folder. Consequently, when importing files from an earlier version, the Task Name column will be ignored.

Custom ETL file

The customEtl.csv file consists of one row per custom ETL (Pre Loading ETL or Post Loading ETL).

<table>
<thead>
<tr>
<th>Header name</th>
<th>Mandatory</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Description</td>
<td>No</td>
<td>Empty</td>
<td>Empty</td>
<td>-</td>
</tr>
<tr>
<td>Type</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Pre Loading ETL or Post Loading ETL</td>
</tr>
<tr>
<td>Active</td>
<td>Yes</td>
<td>False</td>
<td>False</td>
<td>Boolean</td>
</tr>
</tbody>
</table>
## 5 Data Warehouse projects

<table>
<thead>
<tr>
<th>Header name</th>
<th>Mandatory</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sequence Number</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Valid values are positive integer numbers</td>
</tr>
<tr>
<td>Execute as Stored Procedure</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Boolean</td>
</tr>
</tbody>
</table>

- For each custom ETL, Compose will export/import an SQL file to: `<specified export folder>/data mart name>/SQL`
- The file name will be `DM_Custom_ETL_<custom ETL name>.SQL`
- If you wish to edit the file name, make sure that it only contains the following characters: A-Z, 0-9, underscore (_), or space. On import, any other character will be replaced with an underscore.

### Migrating data marts

You can migrate data marts from one environment to another, while preserving custom objects in the target environment.

The following diagram shows a data mart model:

![Data Mart Diagram](image)

### Exporting data marts

When you export data marts, a separate folder containing the CSV files will be created for each data mart:

**Example:**

`datamarts.csv`
Data Warehouse projects

datamart1\facts.csv

datamart1\factdimensions.csv

datamart1\dimensions.csv

datamart1\factattributes.csv

datamart1\dimensionattributes.csv

datamart2\facts.csv

datamart2\factdimensions.csv

datamart2\dimensions.csv

datamart2\factattributes.csv

datamart2\dimensionattributes.csv

Data marts CSV
The datamarts.csv consists of one row per data mart.

<table>
<thead>
<tr>
<th>Header name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Data Mart Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Description</td>
<td>No</td>
<td>Empty</td>
<td>Empty</td>
<td>-</td>
</tr>
<tr>
<td>Table Prefix</td>
<td>No</td>
<td>Empty</td>
<td>Empty</td>
<td>Default for future fact and dimension creation</td>
</tr>
<tr>
<td>View Prefix</td>
<td>No</td>
<td>Empty</td>
<td>Empty</td>
<td>Default for future fact and dimension creation</td>
</tr>
</tbody>
</table>

When exporting a data mart, the following objects will not be included: View schema, database name, and schema name. As these objects are environment-specific, they need to be set up manually after importing the data mart to the target environment (unless you wish to use the defaults from the data warehouse).

Facts CSV
The facts.csv consists of one row per fact table.
## 5 Data Warehouse projects

### Facts CSV metadata mapping

<table>
<thead>
<tr>
<th>Header name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
</table>
| Name               | Yes      | Reject               | Reject                              | Case insensitive  
Currently, the name corresponds to the fact dimension name |
| Description        | No       | Empty                | Empty                               | - |
| Fact Type          | No       | On ADD: Transactional| On ADD: Transactional               | Transactional, Aggregated, or State Oriented |
| Fact Table Name    | Yes      | Reject               | Reject                              | - |
| Fact View Name     | No       | Empty                | Empty                               | Empty means that no view is created |
| Transaction Date   | *No      | If mandatory, reject | If mandatory, reject                 | Mandatory for transactional and aggregated facts  
Ignored for state-oriented facts  
Includes full path with dot notation; for example, in AdventureWorks it might be:  
`OrderDetail.OrderHeader.ModifiedDate` |
| Source Filter      | No       | No filter            | No filter                           | Filter on the source columns (which would eventually translate to an SQL "where" statement)  
Must be formatted as an expression |
| Source Filter Params| No      | Same name for attribute-parameter, similar to expression below | Same name for attribute-parameter, similar to expression below. | - |
| Aggregation Filter | No       | No filter            | No filter                           | Filter on the aggregated columns (which would eventually translate to an SQL "having" statement)  
Must formatted as an expression |
## 5 Data Warehouse projects

<table>
<thead>
<tr>
<th>Aggregation Filter Params</th>
<th>No</th>
<th>Same name for attribute-parameter, like expression below</th>
<th>Same name for attribute-parameter, like expression below</th>
<th>-</th>
</tr>
</thead>
<tbody>
<tr>
<td>Root entity</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>The root entity used. For example, if the fact is a denormalization of order details and orders, it will contain “orders”</td>
</tr>
<tr>
<td>Fact As Type 1</td>
<td>No</td>
<td>Enable the option</td>
<td>Enable the option</td>
<td>Boolean: Accepts the values TRUE or FALSE For an explanation of this option, see Editing star schemas (page 221).</td>
</tr>
</tbody>
</table>

**Dimensions CSV**

The *dimensions.csv* consists of one row per dimension.

### Dimensions CSV metadata mapping

<table>
<thead>
<tr>
<th>Header name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Description</td>
<td>No</td>
<td>Empty</td>
<td>Empty</td>
<td></td>
</tr>
<tr>
<td>Dimension Table Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td></td>
</tr>
<tr>
<td>Dimension View Name</td>
<td>No</td>
<td>Empty</td>
<td>Empty</td>
<td>Empty means that no view is created</td>
</tr>
<tr>
<td>History Type</td>
<td>No</td>
<td>Type 2</td>
<td>Type 2</td>
<td>Type 1/ Type 2</td>
</tr>
<tr>
<td>Source Filter</td>
<td>No</td>
<td>No filter</td>
<td>No filter</td>
<td>Filter on the source columns (which would eventually translate to an SQL &quot;where&quot; statement) Must be formatted as an expression</td>
</tr>
<tr>
<td>Source Filter Params</td>
<td>No</td>
<td>Same name for attribute-parameter, like expression below</td>
<td>Same name for attribute-parameter, like expression below</td>
<td>-</td>
</tr>
</tbody>
</table>
# Data Warehouse projects

<table>
<thead>
<tr>
<th>Header name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Root entity</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>The root entity used. For example, if the fact is a denormalization of order details and orders, it will contain &quot;orders&quot;</td>
</tr>
</tbody>
</table>

**Fact dimensions CSV**
The *factdimensions.csv* consists of one row per dimension usage in the fact dimension.

### Fact dimensions CSV metadata mapping

<table>
<thead>
<tr>
<th>Header name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fact Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Dimension Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
</tbody>
</table>

**Fact attributes CSV**
The *factattributes.csv* consists of one row per attribute and includes the OID attributes as well.

On export, the order is determined by the attributes order. On import, the order is determined by the read order.

### Fact attributes CSV metadata mapping

<table>
<thead>
<tr>
<th>Header name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Star schema Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Attribute Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
</tbody>
</table>
### Data Warehouse projects

<table>
<thead>
<tr>
<th>Header name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If column exists but value is empty</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Entity Path</td>
<td>No</td>
<td>Treat as empty</td>
<td>Treat as empty</td>
<td>Case insensitive. If the column is directly mapped to a data warehouse, the field will contain the model entity path (for example: Orders.Customers). An empty field means that the entity path should be calculated using an expression</td>
</tr>
<tr>
<td>Description</td>
<td>No</td>
<td>Empty</td>
<td>A specific attribute may have empty value as well.</td>
<td>-</td>
</tr>
<tr>
<td>Data Type</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Use combined syntax: <code>Varchar (50)</code> <code>Decimal(10,2)</code></td>
</tr>
<tr>
<td>Aggregation</td>
<td>No</td>
<td>No aggregation columns (this will return an error if the Fact is aggregated).</td>
<td>No aggregation columns.</td>
<td>Empty or <code>SUM/COUNT/MAX/MIN/COUNT_DISTINCT</code></td>
</tr>
<tr>
<td>Expression</td>
<td>No</td>
<td>No expressions in any attribute.</td>
<td>No expression in that attribute.</td>
<td>See Stored objects (page 48).</td>
</tr>
<tr>
<td>Expression Params</td>
<td>No</td>
<td>All attribute-parameter mapping is trivial (same name)</td>
<td>All attribute-parameter mapping is trivial (same name)</td>
<td>See Stored objects (page 48).</td>
</tr>
</tbody>
</table>

### Dimension attributes CSV

The `dimensionattributes.csv` consists of one row per dimension attribute in a fact dimension. These may also include Date or Time dimensions (e.g. Customer.ModifiedDate).

On export, the order is determined by the attributes order. On import, the order is determined by the read order.
### Dimension attributes CSV metadata mapping

<table>
<thead>
<tr>
<th>Header name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If value is missing</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dimension Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Attribute Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Entity Path</td>
<td>No</td>
<td>Treat as empty</td>
<td>Treat as empty</td>
<td>Case insensitive.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>If the column is directly mapped to a data warehouse, the field will contain the model entity path (for example: Orders.Customers)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Am empty field, means that the entity path should be calculated using an expression</td>
</tr>
<tr>
<td>Description</td>
<td>No</td>
<td>Empty</td>
<td>Specific attribute may have empty value as well</td>
<td>-</td>
</tr>
<tr>
<td>Data Type</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Use combined syntax: <code>Varchar(50)</code>, <code>Decimal(10,2)</code></td>
</tr>
<tr>
<td>Expression</td>
<td>No</td>
<td>No expressions in any attribute</td>
<td>No expression in that attribute.</td>
<td>See Stored objects (page 48).</td>
</tr>
<tr>
<td>Expression Params</td>
<td>No</td>
<td>All attribute-parameter mapping is trivial (same name)</td>
<td>All attribute-parameter mapping is trivial (same name)</td>
<td>See Stored objects (page 48).</td>
</tr>
</tbody>
</table>

### Migrating reusable transformations

You can migrate reusable transformations from one environment to another, while preserving custom objects in the target environment.

For information on reusable transformations, see [Defining reusable transformations](page 176).

### Reusable transformations

The `ReusableTransformation.csv` file includes a row per reusable transformation parameter and is described in the table below. Note that some reusable transformations may have no parameters.
5  Data Warehouse projects

<table>
<thead>
<tr>
<th>Header name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If value is missing</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Category</td>
<td>No</td>
<td>On ADD: General</td>
<td>On ADD: General</td>
<td>Case sensitive</td>
</tr>
<tr>
<td>Description</td>
<td>No</td>
<td>Empty</td>
<td>Empty</td>
<td></td>
</tr>
<tr>
<td>Expression</td>
<td>No</td>
<td>On ADD: Empty</td>
<td>On ADD: Empty</td>
<td></td>
</tr>
</tbody>
</table>

**Reusable transformations parameters**

The *ReusableTransformationParams.csv* file is described in the table below.

<table>
<thead>
<tr>
<th>Header name</th>
<th>Required</th>
<th>If column is missing</th>
<th>If value is missing</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reusable Transformation Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Parameter Name</td>
<td>Yes</td>
<td>Reject</td>
<td>Reject</td>
<td>Case insensitive</td>
</tr>
<tr>
<td>Data Type</td>
<td>No</td>
<td>On ADD: VarChar</td>
<td>On ADD: VarChar</td>
<td>According to reusable transformation data types (i.e. no length and scale)</td>
</tr>
<tr>
<td>Description</td>
<td>No</td>
<td>Empty</td>
<td>Empty</td>
<td></td>
</tr>
</tbody>
</table>

**General guidelines**

It's important to take note of the following:

- On export:
  - The order in which the parameters will be written is determined by their order in the web console
  - The order of reusable transformations in the CSV file is alphabetical
  - Reusable transformations are considered part of the model, which means users need the import model permission in order to import them.
  - Column order has no meaning; only column names (case insensitive)
  - All CSVs are optional
  - The CSV files do not contain or rely on internal object IDs, rather, they rely on object names.

- On import, existing reusable transformations will be overridden by the information from the CSV file.

**Commands for exporting/importing objects**

This topic explains how to export and import Compose objects using the CLI. Before performing any import operations, it is strongly recommended to review the topic(s) that discuss CSV file structure and the impact of missing columns and/or values on the target environment. For example, before importing Data Marts, review the Migrating Data Marts topic.
Exporting objects

To export Compose Objects:

1. Open a command prompt and change the working directory to:
   `<COMPOSE_INSTALL_DIR>`\bin>
   Default:
   `C:\Program Files\Qlik\Compose\bin>`
2. Run the following command:
   `ComposeCli.exe connect`
   The following message should be displayed:
   `ComposeControl Program completed successfully.`
3. Then, issue the following command:
   `ComposeCli.exe export_csv --project project_name --outfolder folder`
   Where:
   --project is the name of the source project. This is required.
   --outfolder is the target folder for the CSV files. This is required.

   **Example:**
   
   `ComposeCli.exe export_csv --project myproject --outfolder c:\MyCFDWProject`
   
   When the export completes, the following message should be displayed:
   `ComposeControl Program completed successfully.`

Importing objects

As CSV files do not have versions, Compose cannot know which Compose version the CSV file being imported originated from. If the default behavior has changed between versions, the rule is that the default of the new version will always be applied. For example, in the Compose May 2021 version, the option to update the fact with changes to Type 2 data warehouse entities is now enabled by default. In previous versions, facts were not updated with changes to Type 2 data warehouse entities and there was no option to change this behavior. Therefore, continuing with our example, if you want this option to be disabled by default, you would need to add the column for that setting (**Fact As Type 1**) to the `fact.csv` file and set the value to "FALSE" before importing.

To import all or specific Compose Objects:

1. Open a command prompt and change the working directory to:
   `<COMPOSE_INSTALL_DIR>`\bin>
   Default:
   `C:\Program Files\Qlik\Compose\bin>`
2. Run the following command:
   `ComposeCli.exe connect`
   The following message should be displayed:
   `ComposeControl Program completed successfully.`
3. Run the following command:
ComposeCli.exe import_csv --project project_name --infolder folder [--scope model|mappings|datawarehouse|DatawarehouseTasks|datamarts] [--filetype objecttype] [--infile Filename]

Where:

- --project is the name of the source project. This is required.
- --infolder is the full path to the folder containing the CSV files. This is required.

Starting from Compose August 2021 SR1, the --infolder parameter is only required if you want to import all files, as shown in the example below.

- --scope is an optional parameter that allows non-privileged users to import specific objects. When omitted, all objects will be imported.

Possible values are:

- --scope model - Imports the model objects (Attributes Domain, Entities, Attributes, Relationships, Reusable Transformations, and Reusable Transformation Parameters)
- --scope mappings - Imports the mappings metadata and the actual mappings.
- --scope datawarehouse - Imports the mappings as well as all objects included in the DatawarehouseTasks described below.
- --scope DatawarehouseTasks - Imports the data warehouse tasks and related objects (Tasks, Task Settings, Task Mappings, Task Data Warehouse Tables, Task Custom ETLs, and Custom ETLs)
- --scope datamarts - Imports the data mart objects (Data Marts, Facts, Star Schemas, Fact Attributes, Dimension Attributes, Task Settings, and Custom ETLs)

- --filetype is the object type. This parameter is optional. When omitted, all objects will be imported.

If you need to use this parameter, specify one of the following as the parameter value:

- AttributesDomain
- Entities
- Attributes
- Relationships
- MappingsMetadata
- Mappings
- ReusableTransformation
- ReusableTransformationParams
- DataMarts
- Tasks
- CustomETLs

- --infile is the name of the file to import when using non-default file names. This must be specified together with the --filetype parameter described above.
Example: Import All CSV Files

ComposeCli.exe import_csv --project myproject --infolder c:\composecfdw_csv

Compose August 2021 Example: Import a Specific CSV File with a Custom Name
ComposeCli.exe import_csv --project myproject --infolder c:\composecfdw_csv --filetype AttributesDomain --infile myattributesdomain.csv
When the import completes, the following message should be displayed:
ComposeControl Program completed successfully.

Compose August 2021 SR1 Example: Import a Specific CSV File with a Custom Name
ComposeCli.exe import_csv --project myproject --filetype AttributesDomain --infile c:\myattributesdomain.csv
When the import completes, the following message should be displayed:
ComposeControl Program completed successfully.

Commands for comparing and applying objects

This topic explains how to compare and apply Compose objects using the CLI. Before performing any of these operations, it is strongly recommended to review the topic(s) that discuss CSV file structure and the impact of missing columns and/or values on the target environment. For example, before applying changes to a Model, review the "Migrating the Model" topic.

Comparing objects

Comparing the model to be imported with the existing project model allows you to view and optionally edit the proposed changes before applying them.

When you run the "Compare" command, the structure of the CSV output files will be identical to the export output files, but with the addition of a Change Type and Action columns at the end. Note that as the Apply command currently only supports ADD operations, the only value in the Change Type column will be ADD.

If a column or value was deleted in the source environment, the Action column will contain the value "IGNORE". This tells Compose not to delete the corresponding column/value in the target environment when the Apply command is run. You can delete the "IGNORE" before running the Apply command to force the deletion of the corresponding column/value in the target environment. However, as the Apply command does not currently support DELETE operations, deleting the "IGNORE" before running the Apply command will have no effect.

Compare guidelines:

- Unchanged objects will not be written to the output file(s).
- If there was no changes at all to any of the object types, an empty output file will only be created if you include the --create_files_even_when_no_diff parameter in the command (see below).
- For Boolean fields, values are True/False
- Data type is Compose's logical type.
For model, the attributes writing order is according to the new ordinal in the entity.

Mapping metadata, attributes domain and entity order is alphabetical in the CSV file.

Relationship order is by entity name (alphabetical) and according to their order in the entity.

For mapping, the listed order is:
- Alphabetical for mapping names (e.g. `Map_Orders` follows `Map_Customers`)
- According to target columns (model ordinal) within a mapping.

- Within a mapping, FD (From Date) could be one of the rows.
- Source columns which are not mapped to anything will not be included in the output file.
- If the model is not valid, the Compare command may fail.

To run the Compare command:

1. Open a command prompt and change the working directory to:
   `<COMPOSE_INSTALL_DIR>\bin>
   Default: `C:\Program Files\Qlik\Compose\bin>`
2. Run the following command:
   `ComposeCli.exe connect`
   The following message should be displayed:
   `ComposeControl Program completed successfully.`
3. Run the following command:
   `ComposeCli.exe compare_csv --project project_name --infolder folder --changes_folder folder [--create_files_even_when_no_diff]`
   Where:
   --`project` is the name of the target project. This is required.
   --`infolder` is the folder containing the exported CSV files. This is required.
   --`changes_folder` is the folder to where the change files will be written. The file names will be identical to exported CSV files, but with a _changes suffix (e.g. `entity_changes.csv`).
   --`create_files_even_when_no_diff` is required only if you want a template output file to be created when there are no differences (e.g. an empty `entity_changes.csv` file containing only the header columns). This may be useful, for example, if you wish to manually create a changes file.

**Example:**

```
ComposeCli.exe compare_csv --project ProjectEmpty --infolder "C:\1" --changes_folder "C:\2" --create_files_even_when_no_diff
```

Applying objects

Once you are satisfied with the proposed changes, you can then apply them to your project.

**Apply guidelines:**

- If a column is missing on ADD, then the action will be as described in the topic discussing the object elements.
5 Data Warehouse projects

- By default, all changed object definitions will be added. To filter out rows or columns, edit the outputted CSV file as needed. For example, to only apply changes to a specific Data Mart, delete all of the other Data Marts' rows.
- Any non-standard field name headings, should be renamed in the CSV to the Compose standard.
- Column order is insignificant as the columns will be ordered by name (case insensitive).
- For Boolean fields, accepted values are Yes/No, True/False, 1/0 (case insensitive)
- Data type is Compose's logical type.
- Relationship details override the underlying attributes information (e.g. History Type, Is Key, etc.).

*Attributes that are marked as relationships will be skipped on import as they derive their data type from the relationship.*

- All CSV files are optional in the folder.
- For mappings, the apply operation will fail in the following scenarios:
  - The model is not valid.
  - The target does not exist in the model.
  - The mapping attribute does not exist in the model (after applying *attributes.csv*)
  - The Compose database object does not exist.
- If the source schema, table, view or query does not exist, it will be validated after the Apply operation.

**To run the Apply command:**

1. Open a command prompt and change the working directory to:
   `<COMPOSE_INSTALL_DIR>\bin>`
   Default: `C:\Program Files\Qlik\Compose\bin>`
2. Run the following command:
   `ComposeCli.exe connect`
   The following message should be displayed:
   `ComposeControl Program completed successfully.`
3. Run the following command:
   `ComposeCli.exe apply_csv --project project_name --changes_folder folder [--infile filename] --filetype filetype`
   Where:
   - `--project` is the name of the target project. This is required.
   - `--changes_folder` is the folder containing the change files outputted by the compare command.
   - `--infile filename` -- `--filetype filetype` is the name and type of the file whose changes you wish to apply, should you only want to apply one file.
   Filetype must be one of the following:
   - AttributesDomain
   - Entities
   - Attributes
For example, to apply only attributes, include the following parameters in the command: 
```
--infile attributes_changes.csv --filetype Attributes
```

### Exporting and importing projects using the CLI

**Compose CLI requires Administrator permission. To grant Administrator permission, select “Run as administrator” when opening the command prompt.**

Under normal circumstances, use the deployment options described in *Project deployment* (page 45) to export and import projects. For deployment automation or control by another tool, you can use the command line interface (CLI) to perform such tasks.

**To export or import a project or project configuration, you first need to change the default Master User Password.**

For more information on changing the master user password, see *Changing the master user password* (page 31).

See also: *Moving projects from the test environment to the production environment* (page 80) and *Import/export scenarios - When is a password required?* (page 80)

Before running any command, you must run the *Connecting to Qlik Compose server* (page 76) command.

To get help when using the command line, you can run the Help command. For example, for help about exporting a project, issue the following command:

```
composeCli.exe export_project_repository --help
```

This brings up a list of help parameters.

In this section:

- *Connecting to Qlik Compose server* (page 76)
- *Exporting a project* (page 76)
- *Importing a project* (page 77)
- *Exporting the project configuration* (page 78)
Connecting to Qlik Compose server

Run the `connect` command to establish a connection to the Qlik Compose Server. You must run this command before running any other command:

**Syntax:**

```
ComposeCli.exe connect [--url connection-url]
```

where:

- `url` is the connection URL to the system where the server is running, such as `https://machine.domain/qlikcompose`. This is only required if the server is running on a remote machine.

**Example:**

```
ComposeCli.exe connect --url https://mymachine.mydomain/qlikcompose
Compose Control Program started...
Compose Control Program completed successfully.
```

Exporting a project

You can use the Compose CLI to export a project.

Exported projects includes the following:

- Databases
- Model definitions (entities and attributes)
- Mappings
- Custom ETLs
- Data warehouse tasks
- Data mart definitions

**Existing data warehouse tables and generated tasks are not exported. Notifications and schedules are also not exported as they are considered to be environment-specific.**

**Syntax:**

```
ComposeCli.exe export_project_repository --project project_name --outfile output-file [--is_without_credentials or --without_environment_specifics] [--password password] [--master_user_password master_user_password]
```

where:
5 Data Warehouse projects

- project - The name of the project you want to export.
- outfile - The path to and name of the output file. This file is in JSON format. For example: C:\file.json.
- is_without_credentials - Specifies to export the project settings without the encrypted fields. When importing to a new project, you will need to manually enter the project passwords (in the Compose database connection settings) after the import completes. In addition to eliminating the need to specify a password when exporting or importing the project, the is_without_credentials parameter also allows the project to be used in every Compose installation, regardless of its master user password. It is also useful in the event that you would like to keep the existing passwords in the target environment (e.g. when exporting from a testing environment to an existing project in the production environment).
- password - The password for encrypting the credentials in the exported project. The password > qualifier must be used together with the master_user_password qualifier described below. Use the password > qualifier if you want to encrypt the credentials in the exported project, but do not want the source master password to be used in a different environment. The specified password must be at least 32 characters in length and can either be user-devised or generated using the genpassword utility described in Changing the master user password (page 31).
- master_user_password - The master user password defined for the source machine. This must be used together with the password qualifier. Use the master_user_password > qualifier if you want to encrypt the credentials in the exported project, but do not want the source master password to be used in a different environment. In such a case, when you import the project to an environment that has a different master password, you will only need to specify the password qualifier. For instructions on changing the master user password, see Changing the master user password (page 31). See also: Moving projects from the test environment to the production environment (page 80) and Import/export scenarios - When is a password required? (page 80)

Importing a project

You can use the Compose CLI to import a project. If you import to an existing project, all of the project settings, except the project configuration items will be overridden. For information on the project configuration items, see Exporting the project configuration (page 78).

Imported projects includes the following:

- Databases
- Model definitions (entities and attributes)
- Mappings
- Custom ETLs
- Data warehouse tasks
- Data mart definitions

Syntax:

```
Composecli.exe import_project_repository --project project_name --infile input-file [--password password] [--is_without_credentials] [--override_configuration] [--dont_backup_existing_project] [--autogen]
```

where:
5  Data Warehouse projects

- **project** - is the name of the project you want to import.
- **infile** - specifies the full path to the input file (including the file name). This file is in JSON format. For example: C:\file.json.
- **override_configuration** - overrides the existing project configuration. When importing a project, the default is not to override the existing project configuration.
- **dont_backup_existing_project** - specifies not to back up the existing project. By default, existing projects are backed up to the following location (and automatically restored if the import fails):
  `<product_dir>\data\projects\<project_name>_backup_<timestamp>`
- **is_without_credentials** - specifies to import the project settings without the encrypted fields. In this case, you will need to manually enter the project passwords (in the Compose database connection settings).
- **autogen** - If the project is imported successfully, Compose will:
  - Validate the model and adjust the data warehouse if needed
  - Create the data warehouse tables if they do not exist
  - Validate the data warehouse
  - Adjust the data warehouse if needed. If the "Adjust" cannot be performed automatically, the "autogen" process will be stopped.
  - Generate all data warehouse tasks. If Compose encounters an error while generating a data warehouse task, it will skip the problematic task and continue with the remaining tasks.
  - Create, adjust and generate all data marts

> **If an error is encountered during the "Create" or "Generate" operations, it is recommended to switch to the Compose Console and perform these operations manually. Doing so will generate a more detailed error message and enable you to resolve the issue.**

- **password** the password specified with the **password** qualifier during export. For instructions on changing the master user password, see Changing the master user password (page 31).

For information on validating the data warehouse and generating the task, see Creating and managing the data warehouse (page 177). For information on generating the data mart task, see Creating and managing data marts (page 211).

See also: Moving projects from the test environment to the production environment (page 80) and Import/export scenarios - When is a password required? (page 80)

Exporting the project configuration

You can use the Compose CLI to export the configuration settings of an existing project. This includes database definitions, scheduling jobs, and notifications. This is helpful, for example, when you need to migrate configuration settings from a test environment to the production environment.

For information about migrating projects, see Moving projects from the test environment to the production environment (page 80).
5 Data Warehouse projects

Syntax:
composecli.exe export_project_repository_config --project project_name --outfile output_file
[--is_without_credentials] [--password password] [--master_user_password master_user_password]
where:

- **project** is the name of the project you want to export.
- **outfile** specifies the path to and name of the output file. This file is in JSON format. For example: C:\file.json.
- **is_without_credentials** specifies to export the project configuration without the encrypted fields. When importing to a new project, you will need to manually enter the project passwords (in the Compose database connection settings) after the import completes. In addition to eliminating the need to specify a password when exporting or importing the project, the **is_without_credentials** parameter also allows the project configuration to be used in every Compose installation, regardless of its Master User Password. It is also useful in the event that you would like to keep the existing passwords in the target environment (e.g. when exporting from a testing environment to an existing project in the production environment).
- **password** specifies the password for encrypting the credentials in the exported project configuration. The **password** qualifier must be used together with the **master_user_password** qualifier described below. Use the **password** qualifier if you want to encrypt the credentials in the exported project configuration, but do not want the source master password to be used in a different environment. The specified password must be at least 32 characters in length and can either be user-devised or generated using the genpassword utility described in Changing the master user password (page 31).
- **master_user_password** the master user password defined for the source machine. This must be used together with the **password** qualifier. Use the **master_user_password** qualifier if you want to encrypt the credentials in the exported project configuration, but do not want the source Master User Password to be used in a different environment. In such a case, when you import the project configuration to an environment that has a different Master User Password, you will only need to specify the **password** qualifier.

For instructions on changing the Master User Password, see Changing the master user password (page 31).

See also: Moving projects from the test environment to the production environment (page 80) and Import/export scenarios - When is a password required? (page 80)

Importing the project configuration

You can use the Compose CLI to import the configuration settings of an existing project. This includes database definitions, scheduling jobs, and notifications. This is helpful, for example, when you need to migrate configuration settings from a test environment to the production environment. For information about migrating projects, see Moving projects from the test environment to the production environment (page 80).

**Before you can import the project configuration, you must first run the import_project_repository command described in Importing a project (page 77).**

Syntax:
composecli.exe import_project_repository_config --project project_name --infile input-file [--password password] [--is_without_credentials]
where:

- **project** is the name of the project you want to export.
- **infile** specifies the path to and name of the input file. This file is in JSON format. For example: C:\file.json.
- **is\_without\_credentials** specifies to import the project configuration without the encrypted fields. In this case, you will need to manually enter the project configuration passwords (in the database connection settings).
- **password** the password specified with the **password** qualifier when the project configuration was exported.
  
  For instructions on changing the Master User Password, see Changing the master user password (page 31).
  
  See also: Moving projects from the test environment to the production environment (page 80) and Import/export scenarios - When is a password required? (page 80)

### Moving projects from the test environment to the production environment

After successfully creating and testing projects in the test environment, you now want to move those projects to the production environment. You also need to propagate updates from the testing environment to the production environment as necessary. Although it sounds complicated, moving new and updated projects from the test environment to the production environment is actually quite straightforward, as explained below.

See also Import/export scenarios - When is a password required? (page 80).

> The data source and data warehouse display names must be identical in both the testing and the production environments.

**To perform the initial migration from the testing environment to the production environment:**

1. Export the project from the test environment as described in Exporting a project (page 76).
2. Import the test project to the production environment as described in Importing a project (page 77).
3. Edit the connection settings to point to the production data source and data warehouse.
   
   For more information, see Setting up Landing Zone and Data Source connections (page 124) and Setting up a data warehouse connection (page 104) respectively.
4. Configure notifications and scheduling as needed.
   
   For more information, see Scheduling tasks (page 244) and Notifications (page 246) respectively.

**To propagate updates from the testing environment to the production environment:**

1. Export the project from the test environment as described in Exporting a project (page 76).
2. Import the test project to the production environment as described in Importing a project (page 77).

Import/export scenarios - When is a password required?

The following section describes which of the various export/import scenarios require a password to be specified.
In all scenarios, if you import a project to an existing project, the credentials of the existing projects are preserved (as they are part of the project configuration).

Example 1: Moving a project or project configuration between two Compose machines without retaining the project credentials.

This is useful when importing to a new project that will have different project credentials.

In such a scenario, simply add the `is_without_credentials` parameter to either the export or the import command.

Example 2: Moving a project or project configuration between two Compose machines that have the same Master User Password.

In such a scenario, neither the export command nor the import command need to include a password. If you do not want the source and target projects to have the same credentials (for database connectivity, etc.), then you also need to specify the `is_without_credentials` parameter in either the export or the import command.

Example 3: Moving a project or project configuration between two Compose machines that have a different Master User Password, but without revealing the Master User Password of the source machine.

In such a scenario, the export command must include the `password` and `master_user_password` parameters while the import command must include the `password` parameter. The same password (specified with the `password` parameter) must be used for both export and import.

Example 4: Moving a project or project configuration between two Compose machines that have a different Master User Password.

In such a scenario, the export command does not need to include a password, but the import command should specify the Master User Password of the source machine (using the `password` parameter).
Working with environment variables

Requires Compose August 2021 Service Release 03 or later.

Environment variables allow developers to build more portable expressions, custom ETLs, and Compose configurations, which is especially useful when working with several environments, as is the case with DTAP (Development, Testing, Acceptance and Production). Different environments (for example, development and production) often have environment-specific settings such as database names, schema names, and Replicate task names. Variables allow you to easily move projects between different environments without needing to manually configure the settings for each environment. This is especially useful if many settings are different between environments. For each project, you can use the predefined environment variables or create your own environment variables.

Database and schema name variables are supported with the following objects:

- Reusable transformations
- Custom ETLs (Data warehouse and Data marts)
- Mappings lookups
- Mappings and model expressions
- Data mart settings

In this section:

- Limitations and considerations (page 82)
- Locations and names of predefined variables (page 83)
- Compose environment variables CLI guidelines (page 86)
- Working with predefined variables only (page 86)
- Working with user-defined variables only (page 87)
- Working with both user-defined and predefined variables (page 89)
- Removing environment variables (page 90)
- Manually editing the JSON file (page 91)

Limitations and considerations

- Values for predefined variables will be shown in the Compose user interface, but user-defined variables will be shown in variable format (for example, $$\{my\text{variable}\}$$).
- Variable names are case sensitive and can include alphanumeric characters, underscores, and periods only.
- User-defined variable definitions will not be included in projects exported as JSON files or in projects exported as CSV files. However, user-defined variable names will be included in the relevant places in the exported files.
- See also: Exporting and importing projects using the CLI (page 75) and Migrating objects as CSV files (page 46).
- Exported CSV objects that are associated with predefined environment variables (for example, a mapping database and schema) cannot contain user-defined environment variables.
For instance, this is not allowed:

<table>
<thead>
<tr>
<th>Name</th>
<th>Landing Zone Database</th>
<th>Schema</th>
</tr>
</thead>
<tbody>
<tr>
<td>MyMapping</td>
<td>$${(\text{MyDatabase})}$$</td>
<td>$${(\text{MySchema})}$$</td>
</tr>
</tbody>
</table>

See also: *Migrating objects as CSV files (page 46).*

**Locations and names of predefined variables**

Predefined variables are located in the following places:

**Predefined data warehouse variables**

The following predefined data warehouse connection variables can be set:

- $$\{(\text{database.Data Warehouse.connectionInputModeAdvanced})\}$$ - The value can be "True" or "False". "False" means standard connection mode.
- $$\{(\text{database.Data Warehouse.serverName})\}$$
- $$\{(\text{database.Data Warehouse.port})\}$$
- $$\{(\text{database.Data Warehouse.authenticationMethodWindows})\}$$ - The value can be "True" or "False". "True" means Windows authentication.

- Relevant for Microsoft SQL Server only.

- $$\{(\text{database.Data Warehouse.userName})\}$$
- $$\{(\text{database.Data Warehouse.encryptedPassword})\}$$
- $$\{(\text{database.Data Warehouse.odbcString})\}$$

- Only relevant when the Advanced connection option is set.

- $$\{(\text{database.Data Warehouse.jdbcString})\}$$

- Only relevant when the Advanced connection option is set.

- $$\{(\text{database.Data Warehouse.warehouse})\}$$

- Relevant for Snowflake only.

- $$\{(\text{database.Data Warehouse.database})\}$$
- $$\{(\text{database.Data Warehouse.dataverseSchema})\}$$
- $$\{(\text{database.Data Warehouse.datamartSchema})\}$$

For an explanation of data warehouse settings, see *Setting up a data warehouse connection (page 104)*
Predefined landing zone variables

The following predefined landing zone connection variables can be set:

As there can be several landing zones, the middle part of the variable name (landing1 below) is the landing zone name and should be replaced with the actual name of your Landing Zone.

- `$$\{database.landing1.landing.contentType\}` - Values can be "Full Load Only", "Change Processing", or "Full Load and Change Processing"
- `$$\{database.landing1.landing.designatedBy\}` - The value can be "database" or "schema".
- `$$\{database.landing1.landing.database\}`
- `$$\{database.landing1.landing.schema\}`
- `$$\{database.landing1.landing.errorMartSchema\}`
- `$$\{database.landing1.landing.afterApplyingChanges\}` - Values can be "Delete" (the default) or "Archive".
- `$$\{database.landing1.landing.archiveDatabase\}`

Only relevant if "Archive" is set.

- `$$\{database.landing1.landing.archiveSchema\}`

Only relevant if "Archive" is set.

- `$$\{database.landing1.landing.isAssociatedWithReplicateTask\}` - The value can be "True" or "False".
- `$$\{database.landing1.landing.replicateServer\}`

Only relevant if the **Associate with Replicate task** option is set.

- `$$\{database.landing1.landing.replicateTask\}`

Only relevant if the **Associate with Replicate task** option is set.

- `$$\{database.landing1.source.isSourceDefined\}` - The value can be "True" or "False".
- `$$\{database.landing1.source.connectionInputModeAdvanced\}` - The value can be "True" or "False". "False" means standard connection mode.
- `$$\{database.landing1.source.serverName\}`
- `$$\{database.landing1.source.port\}`
- `$$\{database.landing1.source.authenticationMethodWindows\}` - The value can be "True" or "False". "True" means Windows authentication.
5 Data Warehouse projects

Relevant for Microsoft SQL Server only.

- $$\text{database.landing1.source.userName}$$
- $$\text{database.landing1.source.encryptedPassword}$$
- $$\text{database.landing1.source.odbcString}$$

Only relevant when the Advanced connection option is set.

- $$\text{database.landing1.source.jdbcString}$$

For an explanation of landing zone settings, see Defining landing zones (page 132).

Predefined data mart variables

The following predefined data mart connection variables can be set:

As there can be several data marts, the middle part of the variable name (datamart1 below) is the data mart name and should be replaced with the actual data mart name.

- $$\text{datamart.datamart1.databaseDefault}$$ - The value can be "True" or "False".
- $$\text{datamart.datamart1.database}$$
- $$\text{datamart.datamart1.tablesSchemaDefault}$$ - The value can be "True" or "False".
- $$\text{datamart.datamart1.tablesSchema}$$
- $$\text{datamart.datamart1.viewsSchemaDefault}$$ - The value can be "True" or "False".
- $$\text{datamart.datamart1.viewsSchema}$$

For an explanation of data mart settings, see Modifying data mart settings (page 236).

Predefined lookup variables

The following predefined lookup variables can be set:

As there are usually multiple mappings and columns, the middle part of the variable name (mapping1.column1 below) are the mapping and column names and should be replaced with the actual mapping and column name.

- $$\text{mapping.mapping1.column1.lookup.schema}$$

For an explanation of lookup settings, see Using lookup tables (page 193).
Predefined mappings variables

The following predefined mapping variables can be set:

- `$$\{\text{mapping.mapped1.schema} \}`

For an explanation of mappings settings, see *Editing column mappings* (page 189).

## Compose environment variables CLI guidelines

The procedures below describe how to manage environment variables using the Compose CLI. When using the Compose CLI, make sure to adhere to the following guidelines:

- Before running any `environment_variables` commands, run the `connect` command to establish a connection to the Compose Server. For more information on this command, see *Connecting to Qlik Compose server* (page 76).
- Variable names should be specified in the CLI without dollar signs or curly brackets. So, for example, assuming the name of your landing database is `MyLanding`, `$$\{\text{database.MyLanding.database} \}` should be specified as `database.MyLanding.database`.
- When setting a variable with the Compose CLI, any variable names and values with spaces (both user-defined and predefined) must be specified in quotation marks. This means, for example, that all data warehouse variables must be specified with quotation marks as their names always contain a space (for example, `"database.Data Warehouse.serverName"`).
- When setting a Boolean value, use `--boolVal` instead of `--val`. For example, to set `database.Data warehouse.connectionInputModeAdvanced` to "True", specify: `--var "database.Data warehouse.connectionInputModeAdvanced" --boolVal true`

## Working with predefined variables only

When working with predefined environment variables only, the flow is as follows:

1. In the source environment:
   a. Configure your project as desired.
   b. Run the following command to write the predefined environment variables to a JSON file (replacing `projectName` with the name of your Compose project and `JsonFileLocation` with the full path of your JSON file):
      ```sh
      ComposeCli environment_variables --command writePredefined --project projectName --jsonFile JsonFileLocation
      ```
   c. Create a deployment package or export the project using the Compose CLI.
      See also: *Project deployment* (page 45) and *Exporting a project* (page 76).

2. In the target environment:
   a. Deploy the project if you created a deployment package or import the project if you exported it using the Compose CLI.
      See also: *Project deployment* (page 45) and *Importing a project* (page 77).
   b. Copy the JSON file created in the source environment to your preferred location.
5 Data Warehouse projects

c. Edit the JSON file and replace the variable values with the values you want to propagate to the target environment.
   See also: Working with environment variables (page 82).
d. Run the following command to propagate the JSON file variables to the Compose user interface (replacing projectName with the name of your Compose project and JsonFileLocation with the full path of your edited JSON file):

   ComposeCli environment_variables --command setALL --project projectName --jsonFile JsonFileLocation

   Example:

   ComposeCli environment_variables --command setALL --project analytics --jsonFile C:\compose\compose-variables.json

   Running the setAll command will remove any existing environment variables that are not included in the JSON file.

   You can also set any variable by running the set command. This is especially useful if you do not want the JSON file to include passwords. In this case, you would need to run the following command (shown as an example):

   ComposeCli environment_variables --command set --project MyProject --var encryptedPassword --val g56g56y563%

   When you set a password with the CLI, Compose encrypts the password first and then sets it.

e. Run the following command to apply the predefined environment variables and complete the process (replacing projectName with the name of your Compose project):

   ComposeCli environment_variables --command applyPredefined --project projectName

Working with user-defined variables only

When working with user-defined environment variables only, the flow is as follows:

User-defined variables should have the following format in the Compose user interface:

$$${myVariable}$$$

You can use user-defined variables in the following places:

- Expressions - in model, mappings (column expressions, data quality and filters) or data mart
- Lookup conditions and expressions
- Custom ETLs

To set user-defined variables:
1. In the source environment:
   a. In the Compose user interface, include your variable in an object that supports user-defined variables (for example, a Custom ETL in a data warehouse task or an expression), and save your settings.

   **Example: User-defined variable in a Custom ETL**

   ```sql
   UPDATE "RODIE"."DWH2"."TSTG_Employees"
   SET "Title" = $$\{myVar\}
   WHERE "EmployeeID" < 150
   ```

   b. For user-defined variables, use one of the following methods:

   **Method 1:**
   Set each variable individually by running the following command (replacing `projectName` with the name of your Compose project, `varName` with the variable name, and `value` with the variable value):

   ```bash
   ComposeCli environment_variables --command set --project projectName --var varName --val value
   ```

   **Example:**

   ```bash
   ComposeCli environment_variables --command set --project MyProject --var myVar --val Manager
   ```

   **Method 2:**
   Add the user-defined variables directly to the JSON file as described in *Manually editing the JSON file (page 91)* below.

   c. Generate the task(s) with the user-defined variables.

   d. If you set the user-defined variables with the CLI, run the following command to write the user-defined variables to a JSON file (replacing `projectName` with the name of your Compose project and `JsonFileLocation` with the full path of your JSON file):

   ```bash
   ComposeCli environment_variables --command writeCLISet --project projectName --jsonFile JsonFileLocation
   ```

   e. Create a deployment package or export the project using the Compose CLI.
   See also: *Project deployment (page 45)* and *Exporting a project (page 76).*

2. In the target environment:
   a. Deploy the project if you created a deployment package or import the project if you exported it using the Compose CLI.
   See also: *Project deployment (page 45)* and *Importing a project (page 77).*

   b. Copy the JSON file created in the source environment to your preferred location.

   c. Edit the JSON file and replace the variable values with the values you want to appear in the target environment.
   See also: *Working with environment variables (page 82).*
5 Data Warehouse projects

d. Run the following command to propagate the JSON file variables to the Compose user interface (replacing `projectName` with the name of your Compose project and `JsonFileLocation` with the full path of your edited JSON file):

```bash
Compose Cli environment_variables --command setALL --project projectName --jsonFile JsonFileLocation
```

- **Warning**: Running the `setAll` command will remove any existing environment variables that are not included in the JSON file.

e. Generate the relevant tasks as well.

Working with both user-defined and predefined variables

- **Warning**: Each time Compose writes to the JSON file, it overwrites the existing content. Therefore, when working with both user-defined and predefined variables, you need to specify the path to two different JSON files. For convenience, you can then merge the two files into a single JSON file while taking care to use the format described in Manually editing the JSON file (page 91) below.

When working with both user-defined and predefined environment variables, the flow is as follows:

1. In the source environment:
   a. Configure your project as desired.
   b. Run the following command to write the predefined environment variables to a JSON file (replacing `projectName` with the name of your Compose project and `predefinedJsonFileLocation` with the full path of the JSON file that you want to contain your predefined variables):

   ```bash
   Compose Cli environment_variables --command writePredefined --project projectName --jsonFile predefinedJsonFileLocation
   ```

   c. In the Compose user interface, add user-defined variables to supported objects (for example, a Custom ETL in a data warehouse task), and save your settings.

Example: User-defined variable in a Custom ETL

```sql
UPDATE "ROSIE"."DWH2"."TSTG_Employees"
SET "Title" = $$${myVar}
WHERE "EmployeeID" < 150
```

d. For user-defined variables, use one of the following methods:

**Method 1:**

Set each variable individually by running the following command (replacing `projectName` with the name of your Compose project, `varName` with the variable name, and `value` with the variable value):

```bash
```
5 Data Warehouse projects

ComposeCli environment_variables --command set --project projectName --var varName --val value

Example:

ComposeCli environment_variables --command set --project MyProject --var myVar --val Manager

Method 2:
Add the user-defined variables directly to the JSON file as described in Manually editing the JSON file (page 91) below.

e. Generate the task with the user-defined variable(s).
f. If you set the user-defined variables with the CLI, run the following command to write the user-defined variables to a JSON file (replacing projectName with the name of your Compose project and JsonFileLocation with the full path of the JSON file that you want to contain your user-defined variables):

ComposeCli environment_variables --command writeCLI --project projectName --jsonFile userDefinedJsonFileLocation

g. Create a deployment package or export the project using the Compose CLI.

See also: Project deployment (page 45) and Exporting a project (page 76).

2. In the target environment:

a. Deploy the project if you created a deployment package or import the project if you exported it using the Compose CLI.

See also: Project deployment (page 45) and Importing a project (page 77).
b. Copy the JSON file created in the source environment to your preferred location.
c. Edit the JSON file and replace the variable values with the values you want to appear in the target environment.

See also: Working with environment variables (page 82).
d. Run the following command to propagate the JSON file variables to the Compose user interface (replacing projectName with the name of your Compose project and JsonFileLocation with the full path of your edited JSON file):

ComposeCli environment_variables --command setALL --project projectName --jsonFile JsonFileLocation

Running the setAll command will remove any existing environment variables that are not included in the JSON file.

e. Generate all tasks with user-defined variables.
f. Run the following command to apply the predefined environment variables and complete the process (replacing projectName with the name of your Compose project):

ComposeCli environment_variables --command applyPredefined --project projectName

Removing environment variables

Run the "Remove" command if you set a variable that you no longer want to use in the target environment.
Removing a predefined variable will reset it to its previous value. To prevent errors, make sure when you remove user-defined variables, to also edit or remove the custom ETL or expression where the variable is located.

1. Run the following command:
   
   ```bash
   ComposeCli environment_variables --command remove --var VarName --project projectName
   
   Example:
   
   ComposeCli environment_variables --command remove --var MyVariable --project MyProject
   
   2. Generate any task(s) configured with user-defined variables.

Manually editing the JSON file

Before you propagate the JSON file variables to the Compose user interface in the target environment, you need to edit the file and replace the source variable values with the target variable values. It might also be more convenient to create, edit and maintain the JSON file manually instead of (or in addition to) using the writeCLISet and writePredefined commands described above.

The JSON file is split into two sections: “predefinedVariables” and “userDefinedVariables”. When you edit the JSON file, make sure to put predefined variables in the “predefinedVariables” section and user-defined variables in the “userDefinedVariables” section. In addition, make sure to use standard JSON escaping conventions, as shown in the following example:

```json
{
   "predefined": {
      "databases.Data Warehouse.serverName": "myhostname"
   },
   "userDefined": {
      "variable": "value",
      "var2": "val2"
   }
}
```

Editing and saving the JSON file does not automatically set and apply the variables. To do this, run the ‘SetAll’ and ‘ApplyPredefined’ commands. If the JSON file also contains user-defined variables, you will need to generate the associated tasks as well. See below for details.

Propagating the JSON file variables to the Compose user interface

After you have prepared your JSON file and copied it to your preferred location, perform the following procedure to propagate the variables to the Compose user interface:

1. Run the following command to propagate the JSON file variables to the Compose user interface (replacing `projectName` with the name of your Compose project and `JsonFileLocation` with the full path of your edited JSON file):
   
   ```bash
   ComposeCli environment_variables --command setALL --project projectName --jsonFile JsonFileLocation
   ```
5 Data Warehouse projects

Example:

```plaintext
ComposeCli environment_variables --command setALL --project MyProject --jsonFile C:\composeVariables\myVariables.json
```

*Running the setAll command will remove any existing user-defined environment variables from the user-interface, and reset any modified predefined environment variables that are not included in the JSON file to their previous values.*

2. If the JSON file contains predefined environment variables, run the following command (replacing `projectName` with the name of your Compose project):

```plaintext
ComposeCli environment_variables --command applyPredefined --project projectName
```

3. If the JSON file contains user-defined environment variables, generate the associated task(s).

Generating projects using the CLI

The instruction below explain how to automatically generate projects using the CLI. This can be especially useful when deploying projects between different environments.

On the Compose machine:

1. From the Start menu, open the Compose Command Line console.
2. Run the following command:

```plaintext
ComposeCli.exe connect
```
3. Run the following command:

```plaintext
ComposeCli.exe generate_project --project project_name [--database_already_adjusted]
```
Where:

- `project_name` is the name of your project.
- `--database_already_adjusted` is an optional parameter that should only be included if the data warehouse and data marts were adjusted outside of Compose.

When the command is run, Compose will:

- Validate the model.
- Create any data warehouse tables that do not exist.
- Validate the data warehouse.
- Adjust the data warehouse if needed.

```
- If an Adjust script is needed and `--database_already_adjusted` is included in the command, the script (DDL) will not be run as it is assumed that the user ran it manually.
- If the "Adjust" cannot be performed automatically, the process will be stopped.
```

- Generate all data warehouse tasks.
If Compose encounters an error while generating a data warehouse task, it will skip the problematic task and continue with the remaining tasks.

- Create, adjust and generate all data marts.

Exporting project documentation

Relevant to Data Warehouse projects only.

You can export a project to a zip file for record keeping and sharing offline. The project is exported as HTML files which can be easily printed to PDF using the print toolbar button in the HTML page.

To export the project documentation:

1. Open the project as described in Managing and monitoring projects (page 262).
2. Click the downward arrow to the right of the project name and select Generate Project Documentation from the drop-down menu. A zip file with the name of the project and a timestamp of when the documentation was generated will be created (e.g. MyComposeProject_documentation_03_22_2016__15_01_10.zip). Depending on your browser settings, the file will either be automatically downloaded to your browser’s Downloads folder or you will be prompted to save it.
3. To view the documentation, extract the contents of the zip file and then open the index.html file. A browser tab will open displaying the documentation categories in the left pane.
4. Navigate through the documentation using the tree in the left pane and the breadcrumbs above the documentation.
5 Data Warehouse projects

Viewing and downloading DDL scripts

In the DDL Script Files window, you can view and download the data warehouse DDL script files. By default, Compose executes the Create, Adjust and Drop statements immediately upon user request. However, when the Generate DDL scripts but do not run them option is enabled, Compose will only generate the scripts but not execute them.

DDL scripts must be run from the data warehouse database and schema or data warehouse database and data mart schema, depending on the DDL script and the platform type (for example, in Oracle there are no schemas just the database).

For more information on the Create DDL scripts only option, see Project settings (page 38).

To open the DDL Script Files window:

1. Open your project as described in Managing and monitoring projects (page 262).
2. Click the downward arrow to the right of the project name and select Show DDL Scripts from the drop-down menu. The DDL Script Files window opens.

3. To view a script, select the desired script in the Script Files pane on the left. The script will be displayed on the right.
4. To download a script, select the desired script in the Script Files pane on the left. Then click the download button in the top right of the window.
5. To search for an element in the script, start to type in the search box. All strings that match the search query will be highlighted blue.
   You can navigate between search query matches using the arrows to the right of the search box.
   Use the right and left single arrows to navigate matches sequentially. Use the right and left double arrows to jump to the last and first match respectively.
6. To reset the search, either delete the search query or click the "x" to the right of the search box.

Project versioning
Compose provides built-in project version control using the Git engine. Version control enables Compose developers to commit project revisions to both a local and a remote Git repository. If a mistake is made, Compose developers can easily roll back to earlier versions of the project while minimizing disruption to all team members.

Revisions only store metadata and mapping information. After you revert to a saved revision, you will need to recreate the data warehouse and data mart tables.

Configuring version control settings

To define Version Control Settings:

1. From the project drop-down menu, select Version Control > Settings.
The Version Control Settings - Git window opens.
The Local Commits area shows the local root folder where project revisions are committed. The first time a project revision is committed, Compose creates a JSON file with the current project settings. The <project_name>.json file is archived to a ZIP file (<project_name>_deployment.zip), which is located in a project-specific folder under the source-control folder.
2. To enable commits to a remote Git database, select Enable remote commits and then provide the following information:
   - URL - The address of the remote Git database.
   - User name - Your user name for accessing the remote Git database.
   - Password - Your password for accessing the remote Git database.

Committing projects
You can commit a project using the console or using the CLI:

To commit a project to Version Control using the web console:

1. From the project drop-down menu, select Version Control > Commit.
The Commit - <Project_Name> window opens.
2. Enter a message in the Message box and optionally select the Remote push check box. Note that the Remote push check box will be disabled if the Enable remote commits option described above is not selected.
To commit a project to Version Control using the CLI:

Run the following command from the Compose bin directory:

```bash
composecli.exe commit --project project_name [--message message] [--remote]
```

Where:

- `commit` is the verb.
- `project_name` is the name of the project you want to commit.
- `message` is an optional message to accompany the commit.
- `remote` is required if you want to commit the project to a remote Git repository (see above). By default, the project will be committed locally to `<product_dir>\data\source-control`.

To revert to a saved revision:

1. From the project drop-down menu, select Version Control > Revisions history. The Revision History - `<Project_Name>` window opens. By default, the last 10 revisions are shown. You can change this number by selecting one of the available options from the Show drop-down list.
2. Optionally, use the Search box to find a specific revision.
3. Select the desired revision and then click the Deploy to Revision toolbar button.
4. When prompted to confirm the operation, click Yes. The existing project will be replaced.
5. Click Close to close the Revision History - `<Project_Name>` window.

To download a saved revision:

1. From the project drop-down menu, select Version Control > Revisions history. The Revision History - `<Project_Name>` window opens. By default, the last 10 revisions are shown. You can change this number by selecting one of the available options from the Show drop-down list.
2. Optionally, use the Search box to find a specific revision.
3. Select the desired revision and then click the Download Revision as Package toolbar button. The package will be saved as a ZIP file in your browser's default download location.

Creating a diagnostics package

To assist in troubleshooting esoteric issues, a Qlik Support Engineer may ask you for a diagnostics package. The diagnostics package contains the following information:

- The project "data" directory
- Java logs and workflow logs
- .NET logs
- Deployment package file
To create a diagnostics package:

1. From the Project menu, select Create Diagnostics Package.
2. A zip file in the following format will either be downloaded to your computer or you will be prompted to download it (according to your browser settings):
   \[\text{Compose\_Diagnostics\_<project\_name>\_<timestamp>.zip}\]

5.3 Getting started with Data Warehouse projects

This section provides an overview of the Qlik Compose architecture, familiarizes you with its interface and ends with a short tutorial.

In this section:

- High-level flow (page 97)
- Console elements (page 97)
- Data warehouse project tutorial (page 100)

High-level flow

Setting up data warehouse project typically consists of the following stages (simplified):

1. In Qlik Replicate, define a task that replicates the source tables to a landing zone in the data warehouse.
2. In Compose:
   a. Configure access to your data warehouse.
   b. Configure access to your data sources.
   c. Use the "Discover" option to auto-generate a model from the source tables or import an existing model that was created in ERwin. You can even create the model manually if you prefer.
   d. Once your model is ready, create the data warehouse tables and populate them with the source data.
   e. Create a data mart from the data warehouse tables.
   f. Populate the data mart tables.

See also Introduction (page 15).

Console elements

This section will familiarize you with the elements that comprise the Qlik Compose UI.

To open Qlik Compose, from the Windows Start menu, select All Programs > Qlik Compose > Qlik Compose Console.
Management view

The Qlik Compose Console opens in Management view.

In Management view, you can perform the following tasks:

- Create, edit, and delete projects
  For more information, see Adding and managing data warehouse projects (page 35).
- Access Qlik Compose management options, including:
  - Register and view the product license
  - Manage log levels and cleanup options
  - Manage email settings
  For more information, see Managing Compose (page 343).

Designer view

When you add a new project or open an existing project, the console switches to Designer view. If you are in Monitor view (see below), you can switch back to Designer view by clicking the Designer tab in the top right of the console.

Designer view comprises the following panels:
5 Data Warehouse projects

- **Databases** - Configure access to your source database(s) and data warehouse. For more information, see *Setting up Landing Zone and Data Source connections (page 124)* and *Setting up a data warehouse connection (page 104)* respectively.

- **Model** - Create and edit your model. For more information, see *Creating and managing the model (page 141)*.

- **Data Warehouse** - Create the data warehouse tables, generate the task statements, and run data warehouse tasks. For more information, see *Creating and managing the data warehouse (page 177)*.

- **Data Mart** - Define data marts, create the data mart tables, generate the task statements, and run data mart tasks. For more information, see *Creating and managing data marts (page 211)*.

In Designer view, each of the panels has a bar below the panel name. The bar can be empty, half-filled or completely filled, according to the current configuration status of the panel properties, as follows:

- No fill (gray) - Not configured

![Data Warehouse bar](gray.png)

- Half filled - Configuration is not complete

![Data Warehouse bar](half-filled.png)

- Completely filled - Fully configured

![Data Warehouse bar](fully-filled.png)

**Monitor view**

To switch to Monitor view, click the **Monitor** tab in the top right of the console.
In Monitor view, you can view the status of data warehouse and data mart tasks and schedule their execution, either individually or as a workflow.

For more information, see Controlling and monitoring tasks and workflows (page 240).

Data warehouse project tutorial

This short tutorial will walk you through each of the stages required to create a data warehouse project. For simplicity’s sake, we will be using Microsoft SQL Server as both the source database server and the target Data Warehouse. You can of course use any of the supported source or target databases, but instructions for doing so are outside the scope of this tutorial.

What you need:

- Qlik Compose installed according to the instructions in Qlik Compose installation and setup (page 18).
- The Northwind.MDF sample database attached to Microsoft SQL Server. An easy-to-follow set of instructions for downloading and installing Northwind.MDF can be found at the following website: http://businessimpactinc.com/install-northwind-database/
- Define an empty database on Microsoft SQL Server (e.g. northwind_dwh) and make a note of its name. This will serve as the target Data Warehouse for the Northwind.MDF source tables.
- Microsoft SQL Server Native Client 11.0 installed on the Compose machine.

To set up a Compose project:

1. Define and run a replication task in Qlik Replicate as described in Defining a Qlik Replicate task (page 34).
2. Open Qlik Compose.
3. Add a data warehouse project as described in steps 1-3 of Adding data warehouse projects (page 36).

4. In the Databases panel, perform the following steps to define your data warehouse:
   a. Click Manage. The Manage Databases window opens.
   b. Click the Add New Database link or the New toolbar button. The New Data Warehouse window opens.
   c. In the New Data Warehouse window:
      i. In the Name field, specify a display name for your data warehouse.
      ii. From the Type drop-down list, select Microsoft SQL Server.
      iii. In the Server Name field, specify the Microsoft SQL Server name using the following format:
          - To connect to a named Microsoft SQL Server instance: computer_name\db_server_name
          - To connect to the default Microsoft SQL Server instance: computer_name
      iv. In the User Name and Password fields, enter your credentials for logging in to the server specified in the Server Name field.
      v. In the Database Name field, specify the name of the database specified in the target endpoint of the Qlik Replicate task.
      vi. In the Data Warehouse Schema field, specify dbo or your preferred schema.
      vii. In the Data Mart Schema field, specify dbo or your preferred schema.
          You can specify different schemas for the data warehouse and data mart tables, but for the purpose of this quick start, we’ll use the same schema.
      d. Click Test Connection to verify that Compose is able to establish a connection to the specified database and then click OK to save your settings.
   d. Click New. The New Data Source window opens.
   e. In the New Data Source window:
      i. In the Name field, specify a display name for your data source.
      ii. From the Content Type drop-down list, choose Full Load and Change Processing.
      iii. From the Designate By drop-down list, choose Schema.
      iv. In the Schema name field, enter the schema name that you specified in the Target Metadata tab of the Replicate task. For more information, see Defining a Qlik Replicate task (page 34).
      v. In the Error mart schema name field, specify the schema where you want the data mart exception tables to be created. Data that is rejected by data quality rules will be copied to tables in the specified schema.
      vi. Select the Source database connection check box and then provide the details for connecting to the source database. For the purpose of this tutorial, except for the Schema, these should be the same as the data warehouse connection details.
      vii. Click Test Connection to verify that Compose is able to establish a connection to the
specified database and then click OK to save your settings.

- Click OK to save your settings.

5. In the Model panel, perform the following steps to create the model for data warehouse generation:
   a. From the drop-down menu in the top right corner of the Model panel, select Discover. The Discover window opens.
   b. Select the source database (i.e. the database without the "_landing" suffix). This is the source endpoint in the Qlik Replicate task. The Source Table/View Selection - <Data_Source_Name>_Landing window opens.
   c. In the Source Table/View Selection window:
      - Select the Tables option.
      - Click the Search button.
      - From the Results list, select which tables to discover and then click OK. The Generating Model from <db_name> window opens.
   d. Wait for the model generation to complete and then click Close.

6. In the Data Warehouse panel, perform the following steps to populate the Data Warehouse with the source data:
   a. Click Create. The Creating Data Warehouse window opens. Wait for the Data Warehouse to be created and then click Close.
   b. Click Manage. The Manage Data Warehouse Tasks window opens.
   c. Click Generate. The Generating Statements for Task: <Name> window opens. Wait for the ETL instruction set to be generated and then click Close.
   d. Click Run. The Manage Data Warehouse Tasks window switches to Monitor view and Qlik Compose starts to populate the Data Warehouse with data (this may take a few seconds).
   e. Wait for the Data Warehouse to be populated and then close the Manage Data Warehouse Tasks window.

7. In the Data Mart panel, perform the following steps to create a data mart with a star schema:
   a. Click New. The New Data Mart window opens. Leave the default name.
   b. Make sure the Start New Star Schema Wizard check box is selected, and click OK. The New Star Schema wizard opens. Leave the default name.
   c. Select Transactional as the star schema type and then click Next.
   d. In the Facts screen, select Order Details. Then click Next.
   e. In the Dimensions screen, clear all the check boxes and then select Customers and Products only, as shown below.
f. Then click **Next**.

g. In the **Transaction Date** screen, select **OrderDate** and then click **Finish**. The star schema is displayed on the right of the **Manage Data Marts** window.

<table>
<thead>
<tr>
<th>Data Mart Model</th>
<th>Pre Loading</th>
<th>Post Loading</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>New Schema</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>New Dimension</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Import Dimensions</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Add Dimension to Schema</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Click</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Close</strong></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

h. Click **Create Tables**. The **Creating Data Mart: <Data Mart Name>** window opens. Wait for the Data Mart tables to be created and then close the window.
5.4 Setting up a data warehouse connection

This section explains how to set up data warehouse connectivity in a Qlik Compose project.
5 Data Warehouse projects

The data warehouse contains the landing zone tables (the target of the Qlik Replicate task), the logical entities, the actual data warehouse tables and the data mart tables.

For more information on the data warehouse structure in an project Qlik Compose project, see Introduction (page 15).

Note that Qlik Compose will not let you add data sources before you add a data warehouse. This is because the server connection settings for the source landing zone are derived from the data warehouse settings.

For all supported data warehouse types, each data warehouse schema (or database if there are no schemas) should be used exclusively for a single data warehouse. In other words, using the same schema for different projects, data warehouses and landing zones is not allowed. Data mart schemas, however, can be shared by different data marts.

For more information on adding data sources, see Setting up Landing Zone and Data Source connections (page 124).

For instructions on adding a data warehouse, see the following according to your data warehouse type.

- Using Microsoft SQL Server as a data warehouse (page 105)
- Using Oracle as a data warehouse (page 108)
- Using Amazon Redshift as a data warehouse (page 115)
- Using Microsoft Azure Synapse Analytics as a data warehouse (page 117)
- Using Snowflake as a data warehouse (page 111)
- Using Google Cloud BigQuery as a Data Warehouse (page 120)

Using Microsoft SQL Server as a data warehouse

Although the procedures in this section specifically refer to Microsoft SQL Server, they are equally applicable to Microsoft Azure SQL Managed Instance and Microsoft Azure SQL Database.

It contains the following topics:

- Prerequisites (page 105)
- Working with Windows authentication (page 137)
- Microsoft SQL Server data types (page 106)
- Defining the connection parameters (page 107)

Prerequisites

Before you can use Microsoft SQL Server as a data warehouse in a Qlik Compose project, make sure that the following prerequisites have been met:

Client

Microsoft SQL Server Native Client must be installed on the Qlik Compose machine.
Permissions
To use Microsoft SQL Server as a Data Warehouse a Qlik Compose project, the Compose user must be granted the following privileges in the Microsoft SQL Server database:

- The Qlik Compose user must have at least the db_owner user role on the Microsoft SQL Server database.
- The Qlik Compose user must be granted the CREATE VIEW permission on the Microsoft SQL Server database.
- A Microsoft SQL Server system administrator must provide this permission for all Qlik Compose users.

Working with Windows authentication
You can configure the Qlik Compose Microsoft SQL Server source to log in to Microsoft SQL Server using Windows authentication. If you choose this option, you also need to make sure that:

- The Microsoft SQL Server instance is set up to allow Windows log on.
- The Compose user is specified as the "Log on as" user for the Qlik Compose Server service account.
  OR
  Microsoft SQL Server is configured to allow login for the Qlik Compose Server service account.

Microsoft SQL Server data types
The following table shows the Microsoft SQL Server data warehouse data types that are supported when using Qlik Compose and the default mapping from Qlik Compose data types.

For information on how to view the data type that is mapped from the source, see the section for the source database you are using.

<table>
<thead>
<tr>
<th>Qlik Compose data types</th>
<th>Microsoft SQL Server data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bigint</td>
<td>BIGINT</td>
</tr>
<tr>
<td>Decimal</td>
<td>NUMERIC (p,s)</td>
</tr>
<tr>
<td>Integer</td>
<td>INT</td>
</tr>
<tr>
<td>Date</td>
<td>DATETIME2</td>
</tr>
<tr>
<td>Datetime</td>
<td>DATETIME2</td>
</tr>
<tr>
<td>GUID</td>
<td>UNIQUEIDENTIFIER</td>
</tr>
<tr>
<td>IntAutoInc</td>
<td>INT IDENTITY</td>
</tr>
<tr>
<td>Byte</td>
<td>VARBINARY (Length)</td>
</tr>
</tbody>
</table>
Defining the connection parameters

This section describes how to use a Microsoft SQL Server database as a data warehouse in a Qlik Compose project. You can also use the Microsoft SQL Server data warehouse settings to specify connection details to a Microsoft Azure SQL Database.

When using Microsoft Azure SQL Database as the data warehouse, the data warehouse must be located on the same database as the landing zone, although it should use a different schema.

To define Microsoft SQL Server as a data warehouse:

1. Open your project and click Manage in the bottom left of the Databases panel. The Manage Databases window opens.
2. Click the New toolbar button or click the Add new database link in the middle of the window. The New Data Warehouse dialog box opens.
3. From the Type drop-down list, select the desired data warehouse.
4. Enter the information as described below.
   - **Connection input mode**: Select Standard or Advanced.
     If you selected Standard connection input mode, specify the following:
     - **Server Name**: The name or IP address of the Microsoft SQL Server server machine.
     - **Port**: Optionally, change the default port. When connecting to a named instance using the instance name, either change the port to 0 to use the SQL Server Browser service to redirect.
     - **Windows authentication / SQL Server authentication**: Choose how you want Compose to log in to the Microsoft SQL Server database. If you choose Windows authentication, see Working with Windows Authentication below.
     - **User Name**: The user name for accessing the Microsoft SQL Server database. The specified user must have read/write privileges on the Microsoft SQL Server database.
     - **Password**: The password for accessing the Microsoft SQL Server database.
   If you selected Advanced connection input mode, specify the following:
   - **Password**: The password for accessing the Microsoft SQL Server database.
   - **ODBC Connection String**: The string of parameters required to connect to the Microsoft SQL Server ODBC Driver.
   - **JDBC Connection String**: The string of parameters required to connect to the Microsoft SQL Server JDBC Driver.

Compose will concatenate the database name to the ODBC/JDBC connection string.
5.  **Warehouse Properties**: Specify the following:

- **Database Name**: The name of the Microsoft SQL Server database.
- **Data Warehouse Schema**: The schema in which to create the data warehouse tables.
- **Data Mart Schema**: The schema in which to create the data mart.

5.  Click **Test Connection** to verify that Compose is able to establish a connection with the specified data warehouse.

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

The database is added to the list on the left side of the **Manage Databases** window.

---

**Working with Windows authentication**

You can configure Qlik Compose for Data Warehouses to log in to Microsoft SQL Server using Windows authentication.

If you choose this option, you also need to make sure that:

- The Microsoft SQL Server instance is set up to allow Windows log on.
- The Qlik Compose for Data Warehouses user is specified as the "Log on as" user for the Qlik Compose for Data Warehouses service account.
  -OR-
- Microsoft SQL Server is configured to allow login for the Qlik Compose for Data Warehouses service account.

---

**Using Oracle as a data warehouse**

This section describes how to set up Oracle as a data warehouse in a Compose project.

---

*When loading a huge number of records (i.e. hundreds of millions), the UNDO/REDO/TEMP tablespace on the Oracle database must be large enough to hold the data being loaded.*

---

It contains the following topics:

- **Prerequisites** *(page 108)*
- **Oracle data types** *(page 110)*
- **Defining the connection parameters** *(page 110)*

---

**Prerequisites**

Before you can use Oracle as a data warehouse in a Qlik Compose project, make sure that the following prerequisites have been met:

**Client**

Before you can use Oracle as a source in a Qlik Compose project, make sure that the following client prerequisites have been met:
5 Data Warehouse projects

- The Oracle database should be configured with the required permissions (see below) and accessible from the Compose machine.
- Install Oracle Data Access Components (x64) on the computer where Qlik Compose is located. Then, add the full path of the Oracle Data Access DLL to the system environment variables. The default path should be: 
  `<ORACLE_PRODUCT_CLIENT_DIR>\ODP.NET\bin\4`  
  
  - *The path to the Oracle Data Access DLL also needs to be specified in both the `machine.conf` file and the Global Assembly Cache (GAC). In addition, make sure that the `Oracle.DataAccess.dll` file exists in the following location: `C:\Windows\Microsoft.NET\assembly\GAC_64`.*
  - *The Qlik Compose service needs to be restarted after installing the required components.*

- The Qlik Compose service needs to be restarted after installing the required components.

- Install Oracle Instant Client for Microsoft Windows (x64) on the computer where Qlik Compose is located.
- If you want to use an Oracle TNS name in the connection settings, you first need to set the `ORACLE_HOME` environment variable.
  
  **Example:**
  `<ORACLE_PRODUCT_CLIENT_DIR>\product\12.1.0\client_1`

Permissions

To use Oracle as a Data Warehouse a Qlik Compose project, the Compose user must be granted the following privileges in the Oracle database:

- grant create session
- grant create table
- grant create view
- grant connect
- grant resource
- grant create sequence
- grant create any directory
- grant SELECT on `SYS.DBA_REGISTRY`
- grant select any table
- grant delete any table
- grant drop any table
- grant unlimited tablespace
- grant create any table
- grant insert any table
• grant update any table
• grant drop any table
• grant alter any table
• grant create any view
• grant drop any view
• grant create any index

Oracle data types

The Oracle database for Qlik Compose supports most Oracle data types. The following table shows the Oracle data warehouse data types that are supported when using Qlik Compose and the default mapping from Qlik Compose data types.

For information on how to view the data type that is mapped from the source, see the section for the source database you are using.

<table>
<thead>
<tr>
<th>Qlik Compose data types</th>
<th>Oracle data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bigint</td>
<td>DECIMAL (19,0)</td>
</tr>
<tr>
<td>Date</td>
<td>DATE</td>
</tr>
<tr>
<td>Datetime</td>
<td>DATE</td>
</tr>
<tr>
<td>Decimal</td>
<td>DECIMAL (p,s)</td>
</tr>
<tr>
<td>Integer</td>
<td>DECIMAL (10,0)</td>
</tr>
<tr>
<td>GUID</td>
<td>VARCHAR (38)</td>
</tr>
<tr>
<td>IntAutoInc</td>
<td>DECIMAL (10,0)</td>
</tr>
<tr>
<td>Byte</td>
<td>raw ($[LENGTH])</td>
</tr>
</tbody>
</table>

Defining the connection parameters

This section describes how to use an Oracle database as a data warehouse in a Qlik Compose project.

To define Oracle as a data warehouse:

1. Open your project and click **Manage** in the bottom left of the **Databases** panel. The **Manage Databases** window opens.
2. Click the **New** toolbar button or click the **Add new database link** in the middle of the window. The **New Data Warehouse** dialog box opens.
3. From the **Type** drop-down list, select the desired data warehouse.
4. Enter the information as described below.
   • **Connection input mode**: Select **Standard** or **Advanced**.
     If you selected **Standard** connection input mode, specify the following:
5 Data Warehouse projects

- **Server Name**: The name or IP address of the Oracle server machine.
- **Port**: If you specified a TNS name in the **Server Name** field, make sure this field is empty. Optionally, change the default port.
- **User Name**: The user name for accessing the Oracle database. The specified user must have read/write privileges on the Oracle database.
- **Password**: The password for accessing the Oracle database.
- **SID**: If you specified a TNS name in the **Server Name** field, make sure this field is empty. Optionally, specify the Oracle SID.

If you selected **Advanced** connection input mode, specify the following:
- **Password**: The password for accessing the Oracle database.
- **ODBC Connection String**: The string of parameters required to connect to the Oracle ODBC Driver.
- **JDBC Connection String**: The string of parameters required to connect to the Oracle JDBC Driver.

⚠ Compose will concatenate the database name to the ODBC/JDBC connection string.

- **Warehouse Properties**: Specify the following:
  - **Data Warehouse Schema**: The schema in which to create the data warehouse tables.
  - **Data Mart Schema**: The schema in which to create the data mart.
  - **Maximum length of identifier names is 128**: Select this check box to limit the length of Oracle dimension table identifier names to 128 characters. This should match your Oracle configuration. This option is available for Oracle versions starting from 12.2.

5. Click **Test Connection** to verify that Compose is able to establish a connection with the specified data warehouse.
6. Click **OK** to save your settings.
   The database is added to the list on the left side of the **Manage Databases** window.

Using Snowflake as a data warehouse

This section describes how to set up Snowflake as a data warehouse in a Compose project.

It contains the following topics:

- **Prerequisites** (page 111)
- **Limitations** (page 112)
- **Snowflake data types** (page 113)
- **Defining the connection parameters** (page 113)

Prerequisites

Before you can use Snowflake as a data warehouse in a Qlik Compose project, make sure that the following prerequisites have been met:
Client

- Download and install Snowflake ODBC driver for Windows 2.18.1 or later.
- The Qlik Compose machine must be set to the correct time (UTC).

Permissions

The user specified in the Snowflake data warehouse settings must be associated with a role that grants the following privileges:

- USAGE on the Snowflake warehouse
- USAGE or OWNERSHIP on the specified database and its schemas
- SELECT on the INFORMATION_SCHEMA schemas
- CREATE SCHEMA on the specified database.

Only required for user-specified schemas that do not yet exist on the target.

- Tables:
  - CREATE
  - SELECT
  - INSERT
  - UPDATE
  - DELETE
  - TRUNCATE
  - REFERENCES (for current and future tables)
  - DROP (for user-initiated Drop and Create operations)
- Views:
  - SELECT
  - CREATE (for current and future views)
  - DROP (for user-initiated Drop and Create operations)
- External functions called by user-defined ETLs:
  - USAGE

Limitations

The following limitations apply when using Snowflake as a data warehouse in a Compose project.

- Variant, object and array columns are not supported when creating the model using the discovery method. Compose will ignore such columns during discovery and issue a warning.
- When discovering the landing zone, Snowflake converts all numeric data types to NUMBER (38,0) and is therefore not recommended. For example, discovering a table with INTEGER i and DOUBLE d columns in the landing zone would return NUMBER (38,0) for both, whereas discovering these columns in the source would return more accurate data types.
- When ingesting data from a Replicate source that may have BIT fields (such as Microsoft SQL Server), it is recommended to define a global data type transformation in Replicate to convert BIT to
STRING (1). Otherwise, Compose will convert BIT to VARCHAR (1) (as it does not support BOOLEAN), which may cause a data type mismatch in the landing zone.

Snowflake data types

The following table shows the Snowflake data warehouse data types that are supported when using Qlik Compose and the default mapping from Qlik Compose data types.

For information on how to view the data type that is mapped from the source, see the section for the source database you are using.

<table>
<thead>
<tr>
<th>Qlik Compose data types</th>
<th>Snowflake data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>BIGINT</td>
<td>INTEGER</td>
</tr>
<tr>
<td>INTEGER</td>
<td>INTEGER</td>
</tr>
<tr>
<td>DECIMAL</td>
<td>If column size &gt;38, then:</td>
</tr>
<tr>
<td></td>
<td>DOUBLE</td>
</tr>
<tr>
<td></td>
<td>If column size &lt;38, then:</td>
</tr>
<tr>
<td></td>
<td>DECIMAL (size, scale)</td>
</tr>
<tr>
<td>DATE</td>
<td>DATE</td>
</tr>
<tr>
<td>TIME</td>
<td>TIME</td>
</tr>
<tr>
<td>DATETIME</td>
<td>TIMESTAMP (scale 1-9)</td>
</tr>
<tr>
<td>GUID</td>
<td>VARCHAR (38)</td>
</tr>
<tr>
<td>BIGINTAUTOINC</td>
<td>BIGINT IDENTITY</td>
</tr>
<tr>
<td>BYTE</td>
<td>BINARY</td>
</tr>
</tbody>
</table>

Defining the connection parameters

This section describes how to add Snowflake as a data warehouse in a Qlik Compose project.

To add Snowflake as a data warehouse:

1. Open your project and click Manage in the bottom left of the Databases panel. The Manage Databases window opens.
2. Click the New toolbar button or click the Add new database link in the middle of the window. The New Data Warehouse dialog box opens.
3. From the Type drop-down list, select the desired data warehouse.
4. Enter the information as described below.

   • **Connection input mode**: Select Standard or Advanced.
     If you selected Standard connection input mode, specify the following:
5 Data Warehouse projects

- **Server Name**: The URL for accessing Snowflake on AWS or Snowflake on Microsoft Azure.
- **Port**: The port through which Snowflake will be accessed (default 443).
- **User Name**: The user name for accessing the Snowflake database. The specified user must have read/write privileges on the Snowflake database.
- **Password**: The password for accessing the Snowflake database.

If you selected Advanced connection input mode, specify the following:

- **Password**: The password for accessing the Snowflake database.
- **ODBC Connection String**: The string of parameters required to connect to the Snowflake ODBC Driver.
- **JDBC Connection String**: The string of parameters required to connect to the Snowflake JDBC Driver.

- **Compose will concatenate the warehouse name to the ODBC/JDBC connection string.**

- **You can connect to Snowflake with ODBC, using a proxy server and entering the appropriate ODBC environment parameters. For details, see ODBC Configuration and Connection Parameters - Snowflake Documentation.**

- **You can connect to Snowflake with JDBC, using a proxy server and entering the appropriate JDBC connection string:**

  `jdbc:snowflake://<Snowflake server URL>:443/?&user=<snowflake user name>&warehouse=<Snowflake Warehouse name>&useProxy=true&proxyHost=<Proxy server name>&proxyPort=<Proxy server listening port>&proxyUser=<proxy server user name>&proxyPassword=<proxy server user's password>

- **Warehouse Properties**: Specify the following:
  - **Warehouse Name**: The name of your Snowflake warehouse.

    - "Warehouse" refers to the Snowflake warehouse and should not be confused with the data warehouse created by Compose.

  - **Database Name**: The database in which to create the data warehouse tables.
  - **Data Warehouse Schema**: The schema in which to create the data warehouse tables.
  - **Data Mart Schema**: The schema in which to create the data mart.

5. Click **Test Connection** to verify that Compose is able to establish a connection with the specified data warehouse.
6. Click **OK** to save your settings.
The database is added to the list on the left side of the Manage Databases window.

Using Amazon Redshift as a data warehouse

This section describes how to set up Amazon Redshift as a data warehouse in a Compose project.

It contains the following topics:

- Prerequisites (page 115)
- Amazon Redshift data types (page 116)
- Defining the connection parameters (page 116)

Prerequisites

Before you can use Amazon Redshift as a data warehouse in a Qlik Compose project, make sure that the following prerequisites have been met:

Driver

- Install and configure the latest Amazon Redshift 64-bit ODBC Driver.
- Install the Amazon Redshift JDBC Driver on the Compose machine. Amazon are currently working to resolve a known issue with their latest driver. Until the issue is resolved, JDBC driver 4.1 needs to be used instead.
  1. Download the JAR file from: https://mvnrepository.com/artifact/com.amazon.redshift/redshift-jdbc41/1.2.10.1009
  2. Copy it to: <Compose_Installation_Folder>\java\jdbc

The Qlik Compose service needs to be restarted after copying the driver.

Amazon Redshift Cluster

If you haven't already done so, set up an Amazon Redshift cluster and make sure that following information about your Amazon Redshift Cluster is readily available:

- Amazon Redshift Cluster Name
- Amazon Redshift Cluster Port
- Amazon Redshift User Name and Password
- Amazon Redshift Database Name

Permissions

Qlik Replicate performs the following operations on the replicated tables within Amazon Redshift:

- SELECT, INSERT, UPDATE and DELETE
- Bulk Load
- CREATE, ALTER, DROP
- CREATE VIEW
If the user is the 'DB Owner', these permissions are in place by default. Otherwise, the user must be granted these permissions to achieve successful replication.

Port
Make sure that port 5439 (the Amazon Redshift Cluster port) is open for inbound connections from Qlik Compose.

Amazon Redshift data types
The following table shows the Amazon Redshift data warehouse data types that are supported when using Qlik Compose and the default mapping from Qlik Compose data types.

For information on how to view the data type that is mapped from the source, see the section for the source database you are using.

<table>
<thead>
<tr>
<th>Qlik Compose data types</th>
<th>Amazon Redshift data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>INTEGER</td>
<td>INT4</td>
</tr>
<tr>
<td>BIGINT</td>
<td>INT8</td>
</tr>
<tr>
<td>DECIMAL</td>
<td>NUMERIC (p,s)</td>
</tr>
<tr>
<td>DATE</td>
<td>DATE</td>
</tr>
<tr>
<td>GUID</td>
<td>VARCHAR (38)</td>
</tr>
<tr>
<td>DATETIME</td>
<td>TIMESTAMP</td>
</tr>
<tr>
<td>BYTE</td>
<td>VARCHAR (Length in Bytes)</td>
</tr>
<tr>
<td>TIME</td>
<td>VARCHAR (20)</td>
</tr>
</tbody>
</table>

Defining the connection parameters
This section describes how to use an Amazon Redshift database as a data warehouse in a Qlik Compose project.

To define Amazon Redshift as a data warehouse:

1. Open your project and click Manage in the bottom left of the Databases panel. The Manage Databases window opens.
2. Click the New toolbar button or click the Add new database link in the middle of the window. The New Data Warehouse dialog box opens.
3. From the Type drop-down list, select the desired data warehouse.
4. Enter the information as described below.
   - **Connection input mode**: Select Standard or Advanced.
     If you selected Standard connection input mode, specify the following:
     - **Server Name**: The name or IP address of the Amazon Redshift cluster.
     - **Port**: Optionally, change the default port.
5 Data Warehouse projects

- **User Name**: The user name for accessing the Amazon Redshift database. The specified user must have read/write privileges on the Amazon Redshift database.

- **Password**: The password for accessing the Amazon Redshift database.

If you selected **Advanced** connection input mode, specify the following:

- **Password**: The password for accessing the Amazon Redshift database.

- **ODBC Connection String**: A string of parameters required to connect to the Amazon Redshift ODBC Driver. After entry, click **Test** to verify that a connection was established.

- **JDBC Connection String**: A string of parameters required to connect to the Amazon Redshift JDBC Driver. After entry, click **Test** to verify that a connection was established.

You must include the name of the Amazon Redshift database in the connection string.

- **Warehouse Properties**: Specify the following:
  - **Database Name**: The name of the Amazon Redshift database.
  - **Data Warehouse Schema**: The schema in which to create the data warehouse tables.
  - **Data Mart Schema**: The schema in which to create the data mart.

- **More Options**: Click to see or hide the following advanced options:
  - **Character column size in bytes**: This should be calculated according to the largest value you are likely to store in a `VARCHAR` column. Tables in the Landing Zone will be divided by the specified value (and rounded up).
  
  For example, if the value of **Character column size in bytes** is 3 (the default), both `VARCHAR` (12 bytes) and `VARCHAR` (10 bytes) will be discovered as `VARCHAR` (4 characters).

  See: Use the smallest possible columnsize - Amazon Redshift

  If this value is changed, existing tables will not be affected (i.e. the change will only take effect if new columns are added to the model and the data warehouse tables are updated accordingly).

5. Click **Test Connection** to verify that Compose is able to establish a connection with the specified data warehouse.

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

The database is added to the list on the left side of the **Manage Databases** window.

Using Microsoft Azure Synapse Analytics as a data warehouse

This section describes how to set up Microsoft Azure Synapse Analytics as a data warehouse in a Compose project.
By default, Compose creates tables in Microsoft Azure Synapse Analytics as a CLUSTERED COLUMNSTORE INDEX, which offers the best overall query performance for large tables. Depending on your environment though, you might want to override the default to create all tables or specific tables as a HEAP, for example, which is optimized for smaller tables. For information on how to accomplish this, see Table creation modifiers tab (page 43) in the Project Settings section.

It contains the following topics:

- Prerequisites (page 118)
- Microsoft Azure Synapse Analytics data types (page 118)
- Defining the connection parameters (page 119)

Prerequisites

Before you can use Microsoft Azure Synapse Analytics as a data warehouse in a Qlik Compose project, make sure that the following prerequisites have been met:

Install the Required Client

Install SQL Server Native Client 11 (for connecting to Microsoft Azure Synapse Analytics) on the Compose machine.

Permissions

The user specified in the Microsoft Azure Synapse Analytics connection settings must be granted the following permissions.

Permission required for the specified target database:

The Compose must be granted the db_owner user role on the specified target database.

Permission required for the master database:

The Compose must be granted SELECT access (by adding the user to the master database and then to the db_readers role, for example).

Open the Required Firewall Port(s)

- **When Compose runs on a machine outside Azure** - Open port 1433 for outbound traffic.
- **When Compose runs on an AzureVM** - Open the following ports for outbound traffic:
  - 1433
  - 11000-11999
  - 14000-14999

Microsoft Azure Synapse Analytics data types

The following table shows the Microsoft Azure Synapse Analytics data warehouse data types that are supported when using Qlik Compose and the default mapping from Qlik Compose data types.
For information on how to view the data type that is mapped from the source, see the section for the source database you are using.

### Data type mappings

<table>
<thead>
<tr>
<th>Qlik Compose data types</th>
<th>Microsoft Azure Synapse Analytics data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>BIGINT</td>
<td>BIGINT</td>
</tr>
<tr>
<td>DECIMAL</td>
<td>DECIMAL (p,s)</td>
</tr>
<tr>
<td>INTEGER</td>
<td>INTEGER</td>
</tr>
<tr>
<td>DATE</td>
<td>DATE</td>
</tr>
<tr>
<td>DATETIME</td>
<td>DATETIME2 (s)</td>
</tr>
<tr>
<td>TIME</td>
<td>TIME</td>
</tr>
<tr>
<td>GUID</td>
<td>VARCHAR (38)</td>
</tr>
<tr>
<td>BIGINTAUTOINC</td>
<td>BIGINT</td>
</tr>
<tr>
<td>BYTE</td>
<td>VARBINARY (Length)</td>
</tr>
<tr>
<td>VARCHAR (Length)</td>
<td>VARCHAR (Length)</td>
</tr>
<tr>
<td>NVARCHAR (Length)</td>
<td>NVARCHAR (Length)</td>
</tr>
</tbody>
</table>

### Defining the connection parameters

This section describes how to use a Microsoft Azure Synapse Analytics database as a data warehouse in a Qlik Compose project.

> When using Microsoft Azure Synapse Analytics as the data warehouse, the data warehouse database must be the same as the database that you will later define for the landing zone, although it should use a different schema.

To define Microsoft Azure Synapse Analytics as a data warehouse:

1. Open your project and click Manage in the bottom left of the Databases panel. The Manage Databases window opens.
2. Click the New toolbar button or click the Add new database link in the middle of the window. The New Data Warehouse dialog box opens.
3. From the Type drop-down list, select the desired data warehouse.
4. Enter the information as described below:
   - **Connection input mode**: Select Standard or Advanced. If you selected Standard connection input mode, specify the following:
     - **Server Name**: The name of the Microsoft Azure Synapse Analytics server you are using.
     - **Port**: The port number for the Microsoft Azure Synapse Analytics.
5 Data Warehouse projects

- **User Name**: The user name of a registered Microsoft Azure Synapse Analytics user.
- **Password**: The password for the user entered in the User name field.

If you selected *Advanced* connection input mode, specify the following:

- **Password**: The password for the user entered in the User name field.
- **ODBC Connection String**: The string of parameters required to connect to the Microsoft Azure SQL ODBC Driver.
- **JDBC Connection String**: The string of parameters required to connect to the Microsoft Azure SQL JDBC Driver.

⚠ *Compose will concatenate the database name to the ODBC/JDBC connection string.*

- **Warehouse Properties**: Specify the following:
  - **Database Name**: The name of the target database. This must be the same as the target endpoint defined in the Qlik Replicate task. For more information, see *Defining a Qlik Replicate task (page 34)*.
  - **Data Warehouse Schema**: The schema in which to create the data warehouse tables.
  - **Data Mart Schema**: The schema in which to create the data mart.
- **More Options**: Click to see or hide the following advanced options:
  - **Additional JDBC Parameters**: Any additional parameters you need to add to the default JDBC connection string. The parameters should be separated by a semicolon.
    Format: `PARAM1=VALUE1;PARAM2=VALUE2`
  - **Additional ODBC Parameters**: Any additional parameters you need to add to the default ODBC connection string. The parameters should be separated by a semicolon.
    Format: `PARAM1=VALUE1;PARAM2=VALUE2`

5. Click **Test Connection** to verify that Compose is able to establish a connection with the specified data warehouse.
6. Click **OK** to save your settings.
   The database is added to the list on the left side of the **Manage Databases** window.

**Using Google Cloud BigQuery as a Data Warehouse**

This section describes how to set up Google Cloud BigQuery as a data warehouse in a Compose project.

It contains the following topics:

- **Prerequisites (page 121)**
- **Limitations and Considerations (page 121)**
- **Supported Data Types (page 122)**
- **Setting General Connection Properties (page 123)**
5 Data Warehouse projects

Prerequisites
Before you can use Google Cloud BigQuery as a data warehouse in a Qlik Compose project, make sure the prerequisites described below have been met.

Permissions
When you create your Service Account Key for Google Cloud, make sure to select **BigQuery > BigQuery Data Owner** as the Role. Leave the default key type (JSON) unchanged.

As part of the Service Account Key creation process, a JSON file containing the connection information will be downloaded to your computer. You will need to copy the contents of this file to the **Service account key** field in the Data Warehouse settings.

Client Prerequisites
Both the Simba ODBC driver and the Simba JDBC driver need to be installed on the Compose machine.

To do this:

1. Download the following drivers from [https://cloud.google.com/bigquery/providers/simba-drivers](https://cloud.google.com/bigquery/providers/simba-drivers):
   - Simba ODBC driver 2.3.0 or later (MSI file)
   - Simba JDBC driver 1.2.2 or later (Zip file)
2. To install the ODBC driver, simply run the installer on the Compose machine.
3. To install the JDBC driver, extract all of the files to the `<COMPOSE_INSTALL_DIR>\java\jdbc` folder on the Compose machine.
4. Restart the Compose service.

Limitations and Considerations
The following limitations apply when using Google Cloud BigQuery as a data warehouse in a Compose project.

- The dataset(s) specified in the connection settings must already exist before loading data into Google Cloud BigQuery.
- When ingesting data from a Replicate source that may have BIT columns (such as Microsoft SQL Server), it is recommended to define a global data type transformation in Replicate to convert BIT to STRING (1). Otherwise, Compose will convert BIT to VARCHAR (1) (as it does not support BOOLEAN), which may cause a data type mismatch in the Landing Zone.
- When discovering from a BigQuery landing database, BOOLEAN and FLOAT columns are not supported and will be ignored. If you need such columns to be ingested to the data warehouse, the following workarounds are available:
  - Discover from the source database
  - Convert these data types (which are currently not supported in Compose) to another type such as VARCHAR (1) or INT
- The data warehouse data set and landing data set must be in the same region.
- As strings do not have length in BigQuery, when discovering from the Landing Zone, Compose will assume a default length of VARCHAR(32767). From a practical perspective, since these strings will...
also be created on BigQuery, they will have no runtime length either. To keep things orderly however, best practice is to change strings of known length to their actual expected length.

- Commonly used BigQuery functions were added to the Compose Expression Builder. BigQuery SQL commands that are not listed in the Compose Expression Builder can be entered manually if required.
- BigQuery does not support altering tables via standard DDL operations. To work around this limitation, Compose creates a script that copies the data to a new table. After the data is copied to the new table, make sure to delete the old table.
- Aggregated fact and state oriented data mart are not supported.
- Stored procedures in custom ETLs are not supported.
- Clustering keys are not supported.

Supported Data Types
The following table shows the Google Cloud BigQuery data warehouse data types that are supported when using Qlik Compose and the default mapping from Qlik Compose data types.

For information on how to view the data type that is mapped from the source, see the section for the source database you are using.

<table>
<thead>
<tr>
<th>Compose Data Type</th>
<th>BigQuery Data Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>BYTES</td>
<td>BYTES</td>
</tr>
<tr>
<td>DATE</td>
<td>DATE</td>
</tr>
<tr>
<td>TIME</td>
<td>TIME</td>
</tr>
<tr>
<td>DATETIME</td>
<td>If scale &lt;=6:</td>
</tr>
<tr>
<td></td>
<td>TIMESTAMP</td>
</tr>
<tr>
<td></td>
<td>If scale &gt;6:</td>
</tr>
<tr>
<td></td>
<td>STRING</td>
</tr>
<tr>
<td>BIGINT</td>
<td>INTEGER</td>
</tr>
<tr>
<td>DECIMAL</td>
<td>If the data can be stored in 38,9, then:</td>
</tr>
<tr>
<td></td>
<td>NUMERIC</td>
</tr>
<tr>
<td></td>
<td>If not, then:</td>
</tr>
<tr>
<td></td>
<td>STRING with length of original precision +2</td>
</tr>
<tr>
<td>VARCHAR</td>
<td>STRING</td>
</tr>
<tr>
<td>INTEGER</td>
<td>INTEGER</td>
</tr>
</tbody>
</table>

Unsupported Data Types
The following Google Cloud BigQuery data types are not supported:

GEOGRAPHY, STRUCT, and ARRAY.
Setting General Connection Properties

This section describes how to use a Google Cloud BigQuery database as a data warehouse in a Qlik Compose project.

To define Google Cloud BigQuery as a data warehouse:

1. Open your project and click Manage in the bottom left of the Databases panel. The Manage Databases window opens.
2. Click the New toolbar button or click the Add new database link in the middle of the window. The New Data Warehouse dialog box opens.
3. From the Type drop-down list, select the desired data warehouse.
4. Enter the information as described in the table below.
   - Connection input mode: Select Standard or Advanced.
     If you selected Standard:
     - Paste the contents of the JSON file (including curly brackets) that was downloaded when you created your BigQuery service account key, into the Service account key field.
   If you selected Advanced, specify the following:
     - ODBC Connection String: Enter a string of parameters required to connect to BigQuery via the Simba ODBC Driver.
     - JDBC Connection String: Enter a string of parameters required to connect to BigQuery via the Simba JDBC Driver.

   Compose will concatenate the dataset name to the ODBC/JDBC connection string.

   - Region: Where to upload the dataset created by Compose.
   - Data Warehouse dataset: Specify the dataset in which to create the data warehouse tables.
   - Data mart dataset: Specify the dataset in which to create the data mart.

5. Click Test Connection to verify that Compose is able to establish a connection with the specified data warehouse.
6. Click OK to save your settings.
   The database is added to the list on the left side of the Manage Databases window.

Managing databases

You can edit and delete databases as required. The table below describes the available options.

<table>
<thead>
<tr>
<th>To</th>
<th>Do this</th>
</tr>
</thead>
<tbody>
<tr>
<td>Edit a database</td>
<td>In the left side of the Manage Databases window, select the database that you want to edit and then click the Edit toolbar button.</td>
</tr>
</tbody>
</table>
## 5.5 Setting up Landing Zone and Data Source connections

This section explains how to set up landing zone and data source connectivity in a Qlik Compose project. Note that although you must configure landing zone connectivity, data source connectivity is only required if you want to discover the source database defined in the Replicate task.

For a list of the pros and cons of discovering the source database as opposed to the landing zone, see *Discovering the Source Database or Landing Zone (page 142)*.

### Permissions

This section lists the required permissions for the source landing zone and the source database defined in a Qlik Replicate task.

- *Landing Zone permissions (page 124)*
- *Source database permissions (page 124)*

### Landing Zone permissions

For proper operation, the user specified in the landing zone database connection settings must be granted the following permissions:

- Read metadata
- Select from tables
- Create tables (for error marts)
- Insert to tables (error marts)

For information on the landing zone, see *Landing Zone settings*.

### Source database permissions

To generate the model by discovering the source database in the Replicate task, you need to define a connection to the source database used in the Replicate task. The user defined in the *Source database connection settings* must be granted the following permissions:

- Read metadata (Columns, Primary Keys and Foreign Keys)
- Select from tables

For more information, see *Source Database Connection*. 
Data type mappings

This topic lists the data type mappings from the supported source databases or the supported landing zone databases (where applicable) to the Qlik Compose data types. Note that as MySQL and IBM DB2 for LUW are not supported as data warehouses in Compose, the mappings in those sections are applicable to their role as source databases only.

In this topic:

- Oracle data types (page 125)
- Microsoft SQL Server data types (page 126)
- MySQL data types (page 127)
- Amazon Redshift data types (page 130)
- IBM DB2 for LUW data types (page 130)
- Microsoft Azure Synapse Analytics data types (page 131)
- Google Cloud BigQuery data types (page 132)

Oracle data types

The Oracle database for Qlik Compose supports most Oracle data types. The following table shows the Oracle source data types that are supported when using Qlik Compose and the default mapping to Qlik Compose data types.

For information on how to view the data type that is mapped in the data warehouse, see the section for the data warehouse database you are using.

<table>
<thead>
<tr>
<th>Oracle data types</th>
<th>Qlik Compose data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>CHAR</td>
<td>Varchar</td>
</tr>
<tr>
<td>NCHAR(40)</td>
<td>Varchar(80)</td>
</tr>
<tr>
<td>VARCHAR(2)</td>
<td>VARCHAR</td>
</tr>
<tr>
<td>NUMBER</td>
<td>Decimal</td>
</tr>
<tr>
<td>FLOAT</td>
<td>Decimal(38,12)</td>
</tr>
<tr>
<td>REAL</td>
<td>Decimal(38,12)</td>
</tr>
<tr>
<td>DATE</td>
<td>Date</td>
</tr>
<tr>
<td>TIMESTAMP(6)</td>
<td>Date</td>
</tr>
<tr>
<td>TIMESTAMP(6) WITH LOCAL TIME ZONE</td>
<td>Date</td>
</tr>
<tr>
<td>TIMESTAMP(6) WITH TIME ZONE</td>
<td>Date</td>
</tr>
<tr>
<td>DOUBLE PRECISION</td>
<td>Decimal(38,12)</td>
</tr>
</tbody>
</table>
Non-supported data types

Source Oracle tables with columns of the following Oracle data types are not supported and will be ignored.

- BLOB
- CLOB
- NCLOB
- BFILE
- BINARY_FLOAT
- BINARY_DOUBLE
- INTERVAL YEAR (2) TO MONTH
- INTERVAL DAY (6) TO SECOND (5)
- RAW
- ROWID
- UROWID
- LONG

Microsoft SQL Server data types

The following table shows the Microsoft SQL Server source data types that are supported when using Qlik Compose and the default mapping to Qlik Compose data types.

For information on how to view the data type that is mapped in the data warehouse, see the section for the data warehouse database you are using.

<table>
<thead>
<tr>
<th>Microsoft SQL Server data types</th>
<th>Qlik Compose data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>char</td>
<td>Varchar</td>
</tr>
<tr>
<td>nchar</td>
<td>Varchar</td>
</tr>
<tr>
<td>bit</td>
<td>Integer</td>
</tr>
<tr>
<td>tinyint</td>
<td>Integer</td>
</tr>
<tr>
<td>smallint</td>
<td>Integer</td>
</tr>
<tr>
<td>INT</td>
<td>Integer</td>
</tr>
<tr>
<td>BIGINT</td>
<td>Bigint</td>
</tr>
<tr>
<td>decimal</td>
<td>Decimal</td>
</tr>
<tr>
<td>numeric</td>
<td>Decimal</td>
</tr>
<tr>
<td>smallmoney</td>
<td>Decimal(11,4)</td>
</tr>
<tr>
<td>money</td>
<td>Decimal(20,4)</td>
</tr>
<tr>
<td>float</td>
<td>Decimal(38,12)</td>
</tr>
</tbody>
</table>
### Microsoft SQL Server data types

<table>
<thead>
<tr>
<th>Microsoft SQL Server data types</th>
<th>Qlik Compose data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>real</td>
<td>Decimal(18,6)</td>
</tr>
<tr>
<td>datetime</td>
<td>Date</td>
</tr>
<tr>
<td>datetime2</td>
<td>Date</td>
</tr>
<tr>
<td>smalldatetime</td>
<td>Date</td>
</tr>
<tr>
<td>BINARY</td>
<td>BYTE</td>
</tr>
<tr>
<td>date</td>
<td>Date</td>
</tr>
<tr>
<td>time</td>
<td>Varchar(16)</td>
</tr>
<tr>
<td>uniqueidentifier</td>
<td>GUID</td>
</tr>
</tbody>
</table>

### Non-supported data types

Source Microsoft SQL Server tables with columns of the following Microsoft SQL Server data types are not supported and will be ignored:

- BLOB
- CLOB
- NCLOB
- VARCHAR (MAX)
- TEXT
- NVARCHAR (MAX)
- NVARCHAR (LENGTH)
- NTEXT
- VARBINARY
- IMAGE
- DATETIMEOFFSET
- TIMESTAMP
- SQL_VARIANT
- XML

### MySQL data types

The following table shows the MySQL source data types that are supported when using Qlik Compose and the default mapping to Qlik Compose data types.

For information on how to view the data type that is mapped in the data warehouse, see the section for the data warehouse database you are using.

#### Data type mappings

<table>
<thead>
<tr>
<th>MySQL data types</th>
<th>Qlik Compose data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>BIGINT</td>
<td>Bigint</td>
</tr>
</tbody>
</table>
### MySQL data types

<table>
<thead>
<tr>
<th>MySQL data types</th>
<th>Qlik Compose data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>binary</td>
<td>BYTE</td>
</tr>
<tr>
<td>bit</td>
<td>bigint</td>
</tr>
<tr>
<td>char</td>
<td>Varchar</td>
</tr>
<tr>
<td>date</td>
<td>Date</td>
</tr>
<tr>
<td>datetime</td>
<td>Date</td>
</tr>
<tr>
<td>DECIMAL</td>
<td>Decimal</td>
</tr>
<tr>
<td>double</td>
<td>Decimal(38,12)</td>
</tr>
<tr>
<td>ENUM('x-small', 'small', 'medium', 'large', 'x-large')</td>
<td>Varchar(7)</td>
</tr>
<tr>
<td>FLOAT</td>
<td>Decimal(38,12)</td>
</tr>
<tr>
<td>int</td>
<td>integer</td>
</tr>
<tr>
<td>MEDIUMINT</td>
<td>integer</td>
</tr>
<tr>
<td>MEDIUMTEXT</td>
<td>Varchar(16777215)</td>
</tr>
<tr>
<td>nchar(36)</td>
<td>Varchar(36)</td>
</tr>
<tr>
<td>NUMERIC</td>
<td>Decimal</td>
</tr>
<tr>
<td>REAL</td>
<td>Decimal(38,12)</td>
</tr>
<tr>
<td>set('a','b','c','d')</td>
<td>Varchar(7)</td>
</tr>
<tr>
<td>SMALLINT</td>
<td>integer</td>
</tr>
<tr>
<td>TEXT</td>
<td>Varchar(65535)</td>
</tr>
<tr>
<td>time</td>
<td>Date</td>
</tr>
<tr>
<td>timestamp</td>
<td>Date</td>
</tr>
<tr>
<td>TINYINT</td>
<td>integer</td>
</tr>
<tr>
<td>TINYTEXT</td>
<td>Varchar(255)</td>
</tr>
<tr>
<td>year</td>
<td>integer</td>
</tr>
</tbody>
</table>

### Non-supported data types

Source MySQL tables with columns of the following MySQL data types are not supported and will be ignored:

- GEOMETRY
- GEOMETRYCOLLECTION
- JSON
- linestring
- LONGblob
- LONGTEXT
• mediumblob
• MULTILINestring
• MULTIPoINT
• MULTIPOLYGON
• point
• polygon
• tinyblob
• BIT(64)
• BLOB()
• BIGBLOB
• MEDIUMBLOB
• TINYBLOB
• BLOB
• varbinary (20)

Snowflake data types

The following table shows the Snowflake data types that are supported when using Qlik Compose and the default mapping to Qlik Compose data types.

For information on how to view the data type that is mapped in the data warehouse, see the section for the data warehouse database you are using.

<table>
<thead>
<tr>
<th>Snowflake data types</th>
<th>Qlik Compose data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>NUMBER</td>
<td>DECIMAL</td>
</tr>
<tr>
<td>FLOAT</td>
<td>DECIMAL</td>
</tr>
<tr>
<td>VARCHAR</td>
<td>VARCHAR</td>
</tr>
<tr>
<td>BINARY</td>
<td>BYTE</td>
</tr>
<tr>
<td>BOOLEAN</td>
<td>VARCHAR (5)</td>
</tr>
<tr>
<td>DATE</td>
<td>DATE</td>
</tr>
<tr>
<td>TIME</td>
<td>TIME</td>
</tr>
<tr>
<td>TIMESTAMP_NTZ</td>
<td>DATETIME(9)</td>
</tr>
<tr>
<td>TIMESTAMP_LTZ</td>
<td>DATETIME(9)</td>
</tr>
<tr>
<td>TIMESTAMP_TZ</td>
<td>DATETIME(9)</td>
</tr>
<tr>
<td>VARIANT</td>
<td>JSON or XML as determined by the data source settings.</td>
</tr>
<tr>
<td>OBJECT</td>
<td>N/A</td>
</tr>
<tr>
<td>ARRAY</td>
<td>N/A</td>
</tr>
</tbody>
</table>
Amazon Redshift data types

The following table shows the Amazon Redshift data warehouse data types that are supported when using Qlik Compose and the default mapping to Qlik Compose data types.

For information on how to view the data type that is mapped in the data warehouse, see the section for the data warehouse database you are using.

<table>
<thead>
<tr>
<th>Amazon Redshift data types</th>
<th>Qlik Compose data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>SMALLINT</td>
<td>INTEGER</td>
</tr>
<tr>
<td>INTEGER</td>
<td>INTEGER</td>
</tr>
<tr>
<td>BIGINT</td>
<td>BIGINT</td>
</tr>
<tr>
<td>DECIMAL</td>
<td>DECIMAL</td>
</tr>
<tr>
<td>REAL</td>
<td>DECIMAL (18,6)</td>
</tr>
<tr>
<td>DOUBLE PRECISION</td>
<td>DECIMAL (38,12)</td>
</tr>
<tr>
<td>BOOLEAN</td>
<td>INTEGER</td>
</tr>
<tr>
<td>CHAR</td>
<td>VARCHAR</td>
</tr>
<tr>
<td>DATE</td>
<td>DATE</td>
</tr>
<tr>
<td>TIMESTAMP</td>
<td>DATETIME</td>
</tr>
</tbody>
</table>

IBM DB2 for LUW data types

The following table shows the IBM DB2 for LUW source data types that are supported when using Qlik Compose and the default mapping to Qlik Compose data types.

For information on how to view the data type that is mapped in the data warehouse, see the section for the data warehouse database you are using.

<table>
<thead>
<tr>
<th>IBM DB2 for LUW data types</th>
<th>Qlik Compose data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>DATE</td>
<td>DATE</td>
</tr>
<tr>
<td>TYPE_TIMESTAMP</td>
<td>DATE</td>
</tr>
<tr>
<td>TIMESTAMP</td>
<td>DATE</td>
</tr>
<tr>
<td>TYPE_TIME</td>
<td>DATE</td>
</tr>
<tr>
<td>TYPE_DATE</td>
<td>DATE</td>
</tr>
<tr>
<td>REAL</td>
<td>DECIMAL (18,6)</td>
</tr>
<tr>
<td>DOUBLE</td>
<td>DECIMAL (18,6)</td>
</tr>
<tr>
<td>DECIMAL</td>
<td>DECIMAL</td>
</tr>
</tbody>
</table>
## IBM DB2 for LUW data types vs Qlik Compose data types

<table>
<thead>
<tr>
<th>IBM DB2 for LUW data types</th>
<th>Qlik Compose data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>SMALLINT</td>
<td>INTEGER</td>
</tr>
<tr>
<td>INTEGER</td>
<td>INTEGER</td>
</tr>
<tr>
<td>BIGINT</td>
<td>BIGINT</td>
</tr>
<tr>
<td>WCHAR</td>
<td>VARCHAR</td>
</tr>
<tr>
<td>CHAR</td>
<td>VARCHAR (4000)</td>
</tr>
<tr>
<td>WCHAR</td>
<td>VARCHAR (4000)</td>
</tr>
<tr>
<td>BINARY</td>
<td>BYTE</td>
</tr>
</tbody>
</table>

### Microsoft Azure Synapse Analytics data types

The following table shows the Microsoft Azure Synapse Analytics data warehouse data types that are supported when using Qlik Compose and the default mapping to Qlik Compose data types.

For information on how to view the data type that is mapped to the data warehouse, see the section for the data warehouse you are using.

#### Data type mappings

<table>
<thead>
<tr>
<th>Microsoft Azure Synapse Analytics data types</th>
<th>Qlik Compose data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>DATE</td>
<td>DATE</td>
</tr>
<tr>
<td>DATETIME</td>
<td>DATETIME</td>
</tr>
<tr>
<td>DATETIME2</td>
<td>DATETIME</td>
</tr>
<tr>
<td>SMALLDATETIME</td>
<td>DATETIME</td>
</tr>
<tr>
<td>TIME</td>
<td>TIME</td>
</tr>
<tr>
<td>CHAR</td>
<td>VARCHAR</td>
</tr>
<tr>
<td>NCHAR</td>
<td>VARCHAR</td>
</tr>
<tr>
<td>REAL</td>
<td>DECIMAL (18,6)</td>
</tr>
<tr>
<td>FLOAT</td>
<td>DECIMAL (38,12)</td>
</tr>
<tr>
<td>DECIMAL</td>
<td>DECIMAL</td>
</tr>
<tr>
<td>MONEY</td>
<td>DECIMAL (20,4)</td>
</tr>
<tr>
<td>SMALLMONEY</td>
<td>DECIMAL (11,4)</td>
</tr>
<tr>
<td>BIT</td>
<td>INTEGER</td>
</tr>
<tr>
<td>TINYINT</td>
<td>INTEGER</td>
</tr>
<tr>
<td>SMALLINT</td>
<td>INTEGER</td>
</tr>
<tr>
<td>INT</td>
<td>INTEGER</td>
</tr>
</tbody>
</table>
### Data Warehouse projects

<table>
<thead>
<tr>
<th>Microsoft Azure Synapse Analytics data types</th>
<th>Qlik Compose data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>BIGINT</td>
<td>BIGINT</td>
</tr>
<tr>
<td>VARBINARY</td>
<td>BYTE</td>
</tr>
<tr>
<td>BINARY</td>
<td>BYTE</td>
</tr>
</tbody>
</table>

### Google Cloud BigQuery data types

The following table shows the Google Cloud BigQuery data types that are supported when ingesting data from Google Cloud BigQuery and the default mapping to Qlik Compose data types.

For information on how to view the data type that is mapped in the data warehouse, see the section for the data warehouse database you are using.

#### Data type mappings

<table>
<thead>
<tr>
<th>Google Cloud BigQuery data types</th>
<th>Qlik Compose data types</th>
</tr>
</thead>
<tbody>
<tr>
<td>DATE</td>
<td>Date</td>
</tr>
<tr>
<td>TIME</td>
<td>Time</td>
</tr>
<tr>
<td>TIMESTAMP</td>
<td>Datetime</td>
</tr>
<tr>
<td>INT64</td>
<td>BigInt</td>
</tr>
<tr>
<td>NUMERIC(p,s)</td>
<td>Decimal(p,s)</td>
</tr>
<tr>
<td>FLOAT64</td>
<td>Decimal(38,12)</td>
</tr>
<tr>
<td>STRING</td>
<td>Varchar</td>
</tr>
<tr>
<td>BOOLEAN</td>
<td>Varchar(5) - True or False</td>
</tr>
</tbody>
</table>

### Defining landing zones

In a Compose project, you can define any number of landing zones. Defining multiple landing zones is necessary if the data that you eventually want to be available in your data mart(s) is located in several different landing zones.

Before you can define a landing zone in Qlik Compose, you first need to define a data warehouse.

For more information on adding data warehouses, see *Setting up a data warehouse connection (page 104)*.

#### To add a landing zone:

1. Open your project and click **Manage** in the **Databases** panel. The **Manage Databases** window opens.
2. Click the **New** toolbar button. The **New Data Source** window opens.
3. Provide the following information:
## Data source fields

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Content Type</td>
<td>Choose whether the content in the landing zone is <strong>Full Load, Change Processing</strong> or <strong>Full Load and Change Processing</strong> (according to the Qlik Replicate task definition). See also <strong>After applying changes</strong> below.</td>
</tr>
<tr>
<td>Designated By</td>
<td>Select whether the landing zone is a <strong>Database</strong> or a <strong>Schema</strong>. This should reflect the target endpoint settings in the Qlik Replicate task. When Oracle is the Data Warehouse, this field is read-only since the Oracle landing zone is always designated by <strong>Schema</strong>. For more information, see <strong>Defining a Qlik Replicate task (page 34)</strong>.</td>
</tr>
<tr>
<td>Database Name</td>
<td>This field is not applicable when Oracle is the Data Warehouse. If the landing zone is designated by a <strong>Database</strong>, specify the database name. This must be the same as the target database defined in the Qlik Replicate task. When Microsoft Azure Synapse Analytics is the data warehouse, the landing zone database must be the same as the database defined for the data warehouse, although it should use a different schema. For more information, see <strong>Defining a Qlik Replicate task (page 34)</strong>.</td>
</tr>
<tr>
<td>Schema Name</td>
<td>If a schema name was specified in the Qlik Replicate task settings, specify the same schema name here. When Oracle is the Data Warehouse, this must be the same as the schema defined in the Oracle target connection string in the Qlik Replicate task. For more information, see <strong>Defining a Qlik Replicate task (page 34)</strong>.</td>
</tr>
<tr>
<td>Error Mart Schema Name</td>
<td>Specify the schema where you want the data mart exception tables to be created. Data that is rejected by data quality rules will be copied to tables in the specified schema. For more information on error marts, see <strong>Defining and managing data quality rules (page 198)</strong>.</td>
</tr>
<tr>
<td>After applying changes</td>
<td>If you selected <strong>Change Processing</strong> or <strong>Full Load and Change Processing</strong> as the <strong>Content Type</strong>, you can determine whether the Change Tables will be deleted or archived after the changes have been applied (to the data warehouse tables). If you select <strong>Archive the Change Tables</strong>, you also need to specify a <strong>Database name</strong> and <strong>Schema name</strong> in the relevant fields.</td>
</tr>
<tr>
<td></td>
<td><strong>⚠️</strong> Archived Change Tables do not contain a record of DDL changes. If DDL changes were applied, you will need to update the archived tables manually.</td>
</tr>
</tbody>
</table>
5 Data Warehouse projects

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Discover the VARIANT data type as (applies to Snowflake only)</td>
<td>As Compose does not support mapping directly to the Snowflake VARIANT data type, you need to choose whether VARIANT columns will be created as <strong>JSON</strong> (the default) or <strong>XML</strong> in the Snowflake database.</td>
</tr>
</tbody>
</table>
| Associate with Replicate Task     | Select this to associate your Compose project with the related Replicate task. Replicate tasks replicate the relevant tables from the source database to the landing zone in your data warehouse. Specifying the Replicate task name will enable you to both discover the source tables’ primary keys, and monitor and control that task from within Compose.  

However, before you can specify a Replicate task name, you first need to define the connection settings to at least one Replicate Server machine. To do this, click the **Replicate Server Settings** link below the **Associate with Replicate task** field and then configure the settings as described in **Replicate Server settings (page 134)**.  

Once you have configured connectivity to at least one Replicate Server, you can then proceed to select a Replicate task.  
To select a Replicate task:  
1. Click the browse button to the right of the **Associate with Replicate task** field.  
The **Select Replicate Task** window opens.  
2. Select a Replicate Server from the **Server** drop-down list.  
The **Replicate Tasks** list is populated with all tasks defined on the selected server.  
3. Select the task that is replicating the source tables to the landing zone and then click **OK**.  
The name of the selected task is shown as read-only in the **Associate with Replicate task** field.  

4. If you want to generate the model by discovering the source database in the Replicate task, leave the **New Data Source** window open for now as you will need to define connectivity to the source database in the Replicate task.  
For instructions on how to do this, see **Defining Replicate data source connections (page 134)**.  
Otherwise, click **OK** to save your settings.  

**Defining Replicate data source connections**

You can also generate the model by discovering the source database in the Replicate task. In this case, you will also need to define connectivity to that database.
To define connectivity settings:

1. Open your project and click Manage in the Databases panel. The Manage Databases window opens.
2. Click the New toolbar button. The New Data Source window opens.
3. In the New Data Source window, select the Source database connection option.
4. Continue from one of the following topics as appropriate:
   - Using Oracle as a source (page 135)
   - Using Microsoft SQL Server as a source (page 136)
   - Using MySQL as a source (page 138)
   - Using IBM DB2 for LUW as a source (page 140)

Using Oracle as a source

This section describes how to set up connectivity to the Oracle database defined for the Replicate task. This is required if you want to discover the tables and/or views from the source database as opposed to the landing zone. For a list of the pros and cons of each method, see Discovering the Source Database or Landing Zone (page 142).

It contains the following topics:

- Prerequisites (page 135)
- Oracle data types (page 125)
- Defining the connection parameters (page 136)

Prerequisites

Before you can use Oracle as a source in a Qlik Compose project, make sure that the following prerequisites have been met:

- The Oracle database should be configured with the required Permissions (page 124) and accessible from the Compose machine.
- Install Oracle Data Access Components (x64) on the computer where Qlik Compose is located. Then, add the full path of the Oracle Data Access DLL to the system environment variables. The default path should be: `<ORACLE_PRODUCT_CLIENT_DIR>\ODP.NET\bin\x64`.
- The path to the Oracle Data Access DLL also needs to be specified in both the `machine.conf` file and the Global Assembly Cache (GAC). In addition, make sure that the `Oracle.DataAccess.dll` file exists in the following location: `C:\Windows\Microsoft.NET\assembly\GAC_64`.
- Install Oracle Instant Client 11.2.0.3.0 or later (Windows x64) on the computer where Qlik Compose is located.
- The Qlik Compose service needs to be restarted after installing the required components.
If you want to use an Oracle TNS name in the connection settings, you first need to set the ORACLE_HOME environment variable.

Example:

`<ORACLE_PRODUCT_CLIENT_DIR>\product\12.1.0\client_1`

Defining the connection parameters

You can add an Oracle database to Qlik Compose to use as a source.

To add an Oracle source database to Qlik Compose:

1. In the New Data Source window, enter the information as described in the table below.

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>Select Oracle.</td>
</tr>
<tr>
<td>Server Name</td>
<td>Specify the name or IP address of the Oracle server machine. OR Specify the TNS name.</td>
</tr>
<tr>
<td>Port</td>
<td>If you specified a TNS name in the Server Name field, make sure that this field is empty. Optionally, change the default port.</td>
</tr>
<tr>
<td>User Name</td>
<td>Specify your user name for accessing the Oracle database. The specified user must have read/write privileges on the Oracle database.</td>
</tr>
<tr>
<td>Password</td>
<td>Specify your password for accessing the Oracle database.</td>
</tr>
<tr>
<td>SID</td>
<td>If you specified a TNS name in the Server Name field, make sure that this field is empty. Otherwise, specify the Oracle SID.</td>
</tr>
<tr>
<td>Schema</td>
<td>Specify the schema containing the source tables.</td>
</tr>
</tbody>
</table>

2. Click Test Connection to verify that Compose is able to establish a connection with the specified database.

3. Click OK to save your settings.

The database is added to the list on the left side of the Manage Databases window.

Using Microsoft SQL Server as a source

This section describes how to set up connectivity to the Microsoft SQL Server database defined as the source endpoint for the Replicate task. This is required if you want to discover the tables and/or views from the source database as opposed to the landing zone. For a list of the pros and cons of each method, see Discovering the Source Database or Landing Zone (page 142).

It contains the following topics:
Prerequisites
Before you can use Microsoft SQL Server as a source in a Qlik Compose project, make sure that the following prerequisites have been met:

- Microsoft SQL Server should be configured with the required Permissions (page 124) and accessible from the Compose machine.
- Microsoft SQL Server Native Client must be installed on the Qlik Compose machine.
- Qlik Compose supports the following Microsoft SQL Server editions.
  - Enterprise Edition
  - Standard Edition
  - Workgroup Edition
  - Developer Edition

Working with Windows authentication
You can configure the Qlik Compose Microsoft SQL Server source to log in to Microsoft SQL Server using Windows authentication. If you choose this option, you also need to make sure that:

- The Microsoft SQL Server instance is set up to allow Windows log on.
- The Compose user is specified as the "Log on as" user for the Qlik Compose Server service account.
  OR
- Microsoft SQL Server is configured to allow login for the Qlik Compose Server service account.

Defining the connection parameters
You can add a Microsoft SQL Server database to Qlik Compose to use as a source. You can also use the Microsoft SQL Server source to specify connection details to a Microsoft Azure SQL Database.

When using Microsoft Azure SQL Database as the data warehouse, the data warehouse database must be the same as the database that you will later define for the landing zone, although it should use a different schema.
To add a Microsoft SQL Server source database to Qlik Compose:

1. In the **New Data Source** window, enter the information as described in the table below.

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>Select <strong>Microsoft SQL Server</strong>.</td>
</tr>
<tr>
<td>Server Name</td>
<td>Specify the name or IP address of the Microsoft SQL Server machine.</td>
</tr>
<tr>
<td>Port</td>
<td>Optionally, change the default port.</td>
</tr>
<tr>
<td>Windows authentication</td>
<td>Choose how you want Compose to log in to the Microsoft SQL Server database. If you choose <strong>Windows authentication</strong>, see Working with Windows authentication below.</td>
</tr>
<tr>
<td>SQL Server authentication</td>
<td></td>
</tr>
<tr>
<td>User Name</td>
<td>Specify your user name for accessing the Microsoft SQL Server database. The specified user must have read/write privileges on the Microsoft SQL Server database.</td>
</tr>
<tr>
<td>Password</td>
<td>Specify your password for accessing the Microsoft SQL Server database.</td>
</tr>
<tr>
<td>Database Name</td>
<td>Specify the name of the Microsoft SQL Server database.</td>
</tr>
<tr>
<td>Schema</td>
<td>Specify the schema containing the source tables.</td>
</tr>
</tbody>
</table>

2. Click **Test Connection** to verify that Compose is able to establish a connection with the specified database and/or landing zone.

3. Click **OK** to save your settings.
   The database is added to the list on the left side of the **Manage Databases** window.

**Working with Windows authentication**

You can configure Qlik Compose for Data Warehouses to log in to Microsoft SQL Server using Windows authentication.

If you choose this option, you also need to make sure that:

- The Microsoft SQL Server instance is set up to allow Windows log on.
- The Qlik Compose for Data Warehouses user is specified as the "Log on as" user for the Qlik Compose for Data Warehouses service account.
  - OR -
  - Microsoft SQL Server is configured to allow login for the Qlik Compose for Data Warehouses service account.

**Using MySQL as a source**

This section describes how to set up connectivity to the MySQL database defined as the source endpoint for the Replicate task. This is required if you want to discover the tables and/or views from the source database as opposed to the landing zone. For a list of the pros and cons of each method, see Discovering the Source Database or Landing Zone (page 142).

It contains the following topics:
Prerequisites

Before you can use MySQL as a source in a Qlik Compose project, make sure that the following prerequisites have been met:

- The MySQL database should be configured with the required Permissions (page 124) and accessible from the Compose machine.
  The following MySQL editions are supported:
  - MySQL Community Edition
  - MySQL Standard Edition
  - MySQL Enterprise Edition
  - MySQL Cluster Carrier Grade Edition
- MySQL ODBC 64-bit client must be installed on the same computer as Qlik Compose.

Cluster prerequisites

To be able to discover clustered (NDB) tables, the following parameters must be configured in the MySQL my.ini (Windows) file.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
</table>
| ndb_log_bin              | Must be:
  ndb_log_bin=on
  This ensures that changes in clustered tables will be logged to the binary log. |
| ndb_log_update_as_write  | Must be:
  ndb_log_update_as_write=OFF
  This prevents writing UPDATEs as INSERTs in the binary log. |
| ndb_log_updated_only     | Must be:
  ndb_log_updated_only=OFF
  Ensures that the binary log will contain the entire row and not just the changed columns. |

Defining the connection parameters

You can add a MySQL database to Qlik Compose to use as a source.
To add a MySQL source database to Qlik Compose:

1. In the **New Data Source** window, enter the information as described in the table below.

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>Select <strong>MySQL</strong>.</td>
</tr>
<tr>
<td>Server Name</td>
<td>Specify the name or IP address of the MySQL server machine.</td>
</tr>
<tr>
<td>Port</td>
<td>Optionally, change the default port.</td>
</tr>
<tr>
<td>User Name</td>
<td>Specify your username for accessing the MySQL database.</td>
</tr>
<tr>
<td></td>
<td>The specified user must have read/write privileges on the MySQL database.</td>
</tr>
<tr>
<td>Password</td>
<td>Specify your password for accessing the MySQL database.</td>
</tr>
<tr>
<td>Database Name</td>
<td>Specify the name of the MySQL database.</td>
</tr>
<tr>
<td>Schema</td>
<td>Specify the schema containing the source tables.</td>
</tr>
</tbody>
</table>

2. Click **Test Connection** to verify that Compose is able to establish a connection with the specified database and/or landing zone.

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

The database is added to the list on the left side of the **Manage Databases** window.

Using IBM DB2 for LUW as a source

This section describes how to set up connectivity to the IBM DB2 for LUW database defined as the source endpoint for the Replicate task. This is required if you want to discover the tables and/or views from the source database as opposed to the landing zone. For a list of the pros and cons of each method, see *Discovering the Source Database or Landing Zone (page 142)*.

It contains the following topics:

- Prerequisites (page 140)
- IBM DB2 for LUW data types (page 130)
- Defining the connection parameters (page 140)

Prerequisites

Before you begin to work with an IBM DB2 for LUW database as a source in Qlik Compose, make sure the following prerequisites have been met:

- The IBM DB2 for LUW database should be configured with the required *Permissions (page 124)* and accessible from the Compose machine.
- The IBM Data Server Driver for ODBC and CLI version 10.5 must be installed on the Qlik Compose machine.

Defining the connection parameters

You can add an IBM DB2 for LUW database to Qlik Compose to use as a source.
To add an IBM DB2 for LUW source database to Qlik Compose:

1. In the New Data Source window, enter the information as described in the table below.

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>Select IBM DB2 for LUW.</td>
</tr>
<tr>
<td>Server Name</td>
<td>Specify the name or IP address of the IBM DB2 for LUW server machine.</td>
</tr>
<tr>
<td>Port</td>
<td>Optionally, change the default port.</td>
</tr>
<tr>
<td>User Name</td>
<td>Specify your username for accessing the IBM DB2 for LUW database. The specified user must have read/write privileges on the IBM DB2 for LUW database.</td>
</tr>
<tr>
<td>Password</td>
<td>Specify your password for accessing the IBM DB2 for LUW database.</td>
</tr>
<tr>
<td>Database Name</td>
<td>Specify the name of the IBM DB2 for LUW database.</td>
</tr>
<tr>
<td>Schema</td>
<td>Specify the schema containing the source tables.</td>
</tr>
</tbody>
</table>

2. Click Test Connection to verify that Compose is able to establish a connection with the specified database and/or landing zone.

3. Click OK to save your settings.

The database is added to the list on the left side of the Manage Databases window.

Managing databases

You can edit and delete databases as required. The table below describes the available options:

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
<tbody>
<tr>
<td>Edit a database</td>
<td>In the left side of the Manage Databases window, select the database that you want to edit and then click the Edit toolbar button.</td>
</tr>
<tr>
<td>Delete a database</td>
<td>In the left side of the Manage Databases window, select the database that you want to delete and then click the Delete toolbar button.</td>
</tr>
</tbody>
</table>

5.6 Creating and managing the model

This section describes how to create, import and manage the model.

The model serves as the basis for data warehouse generation in Compose. There are three way of creating the model: Use Compose to derive a tentative model by reverse engineering the source database(s) (a process also known as "discovering"); Import a model created in ERwin or create the model manually in Compose.

In this section:
5 Data Warehouse projects

- Generating the model (page 142)
- Model limitations (page 149)
- Validating the model (page 149)
- Displaying the model (page 150)
- Managing the model (page 153)
- Creating expressions (page 168)
- Opening the expression builder (page 169)
- Defining reusable transformations (page 176)

Generating the model

This section explains how to generate a Business Model from a source database. You can generate the model using any of the following methods:

- Use Compose to discover the source database or landing zone
- Import an ERwin model into Compose
- Create the model manually in Compose

For information about importing a model created in ERwin, see Importing the model from ERwin (page 147).

Discovering the Source Database or Landing Zone

Discovery can either be performed on the source database defined for the Qlik Replicate task or in the landing zone. The decision where to perform the discovery is determined by several factors, as explained in the following table:

<table>
<thead>
<tr>
<th>Factor</th>
<th>Discove the Landing Zone</th>
<th>Discover the Source Database defined for the Qlik Replicate task</th>
</tr>
</thead>
<tbody>
<tr>
<td>The source tables selected in the Qlik Replicate task contain foreign keys that you want to maintain in the Compose project.</td>
<td>-</td>
<td>✓</td>
</tr>
<tr>
<td><strong>Qlik Replicate does not support foreign key replication.</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>The source database defined for the Qlik Replicate task is not natively supported by Qlik Compose.</td>
<td>✓</td>
<td>-</td>
</tr>
<tr>
<td>The selected source tables contain keys that are not relevant to the data warehouse (e.g. surrogate keys and business keys)</td>
<td>✓</td>
<td>-</td>
</tr>
</tbody>
</table>
## 5 Data Warehouse projects

<table>
<thead>
<tr>
<th>Factor</th>
<th>Discover the Landing Zone</th>
<th>Discover the Source Database defined for the Qlik Replicate task</th>
</tr>
</thead>
<tbody>
<tr>
<td>A transformation defined for the Qlik Replicate task means that not all of the columns will be replicated to the landing zone. In this case, you should discover the landing zone since this is the data that you eventually want to appear in your data warehouse.</td>
<td>✓</td>
<td>-</td>
</tr>
</tbody>
</table>

**To generate the model by discovery:**

1. Open your project.
2. To generate the model from within Compose:
   1. In the **Model** panel, select **Discover** from the drop-down menu in the top right corner.
      OR
   2. In the **Manage Model** window, click the **Discover** toolbar button.
      The **Discover** window opens.
3. Select whether to discover the source database or the landing zone and then click **OK**.
   Note that the suffix ".landing" denotes the landing zone whereas the actual source database appears without the suffix.

> **When discovering directly from Microsoft SQL Server source, TIME will be discovered as STRING(16). As well as being mapped this way in Replicate, this will also maintain accuracy when a TIME column is defined with high precision.**
3. Choose one of the following Search for options:
   - To list tables only, select Tables.
   - To list views only, select Views.
   - To list tables and views, select All.

4. If you also want the internal Qlik tables to be included in the search results, select the Show Internal Qlik Tables check box. This may be useful for debugging, but is not usually not necessary.

5. To display all tables/views, click Search.

6. To only display tables/views whose names contain a specific string, type the string in the Name field and then click Search.
   The tables/views will be displayed in the Results list.

7. In the Results list, select the source tables and/or views on which to base the model or click the >> button (Add All) to add all of the tables in the schema.

   You can select multiple tables/views by holding down the [Shift] (sequential selection) or [Ctrl] (non-sequential selection) button.

8. To add the selected tables/views, click the > (Add) button.

   If you add a table that already exists in the model with the same name, then the new table is added with the name: source_table_name_01 (or source_table_name_02 if the name source_table_name_01 already exists, and so on).
   If the table contains attribute domains that differ from existing ones but have the same name, they will also be appended with the _01 suffix.

9. Click OK to generate the model from the selected tables/views.
The **Generating Model from [model name]** window opens. A progress bar indicates the current model generation progress. For each stage of the model generation process, a corresponding message appears in the **Messages** list.

10. After the model has been generated, click **Close**.

11. Repeat Steps 2-9 to discover additional sources.

### Clearing the Landing Zone metadata cache

To improve performance when reading from the Landing Zone or from the Data Warehouse tables, Compose caches the metadata from both the Landing Zone and the Data Warehouse tables. However, synchronization issues may sometimes occur if the metadata structure of the Landing Zone or the Data Warehouse tables is altered outside of the Compose project.

If you aware of external changes to the metadata or if you notice any data synchronization anomalies, Compose enables you to clear the metadata cache, either using the UI or using the CLI.

You can clear the Landing Zone metadata cache using either the Compose web console or the Compose CLI.

**Clearing the metadata cache using the web console**

When using the web console to clear the metadata cache, the following methods are available:

**Method 1:**

1. Open the **Source Table/View Selection - <Landing_Zone_Name>** window as described in *Discovering the Source Database or Landing Zone (page 142).*
2. Click the **Clear Cache** button located below the **Show Internal Qlik tables** option.

**Method 2:**

1. Click the **Manage** button at the bottom left of the **Data Warehouse** panel. The **Manage Data Warehouse Tasks** window opens.
2. In the Mappings tab, click **Clear Landing Cache**.

For information on clearing the Data Warehouse cache, see *Clearing the data warehouse metadata cache (page 210).*

### Clearing the metadata cache using the CLI

**To clear the metadata cache using the CLI:**

1. From the **Start** menu, open the **Compose Command Line** console.
2. Run the following command:
   
   ```bash
   ComposeCli.exe connect
   ```
3. Run the following command:

   ```bash
   ComposeCli.exe clear_cache --project project_name [-type landing|storage] [--landing_zone source_name]
   ```
Where:

- **--project** is the name of your project.
- **--type** is where to clear the metadata cache. Possible values are **landing** or **storage**. If **--type landing** and you want to clear a specific landing zone, you must set the **--landing_zone** parameter as well. To clear the metadata cache in all landing zones, specify **--type landing** and omit the **--landing_zone** parameter.
- **--landing_zone** is the name of the landing zone when **--type landing**.

**Example:**

```
ComposeCli.exe clear_cache --project myproject --type landing --landing_zone mysource1
```

Importing entities and mappings from another project

You can import entities and mappings from another project with the same data warehouse type. This is especially useful within a development environment if you need to integrate a private developer’s project with the main project.

**To import entities and mappings:**

1. Open the **Manage Model** window as described in *Managing the model (page 153)*.
2. In the **Entities** toolbar, click the **Import from Project** button.
3. The **Import from Project** wizard opens.
4. In the **Entities** tab:
   a. Select a project from the **Import from Project** drop-down list.
   b. Optionally, search for specific entities.
   c. Select which entities to import or select **Select All** to import all entities.
5. Click **Next** to select which mappings to import.

```
To create new entities and mappings if the selected entities and mappings already exist, clear the **Replace existing entities and mappings** check box. The new entities/mappings will be named `<existing_name>IMPORTED` (or `<existing_name>IMPORTED_<n>` if the entity/mapping is imported more than once).
```

6. In the **Mappings** tab:
   Either click **Finish** to import all mappings for the selected entities (the default).
   OR
   Select which mappings you want to import and then click **Finish** to import the selected entities and mappings.

```
If you do not wish to import any mappings, clear the **Mappings** check box before clicking **Finish**.
```
Importing the model from ERwin

In order to import a model created in ERwin, you first need to export the model from ERwin to an XML file and then copy the XML file to the Compose Server machine. Note that when you import a model from ERwin, you need to create the Mappings ETL scripts manually. You can either do this by creating global mapping as described in Managing global mappings (page 147) below or you can create the mapping ETL directly in the Data Warehouse panel.

For more information on creating the ETL mapping(s) in the Data Warehouse panel, see Creating and managing the data warehouse (page 177).

To import the model from ERwin:

1. Open your project.
2. To import a model created in ERwin, in the Model panel, select Import from ERwin from the drop-down menu in the top right corner.
   OR
   In the Manage Model window, select Import from ERwin from the Entities drop-down menu.
   The Import from ERwin window opens.
3. Specify the full path to the ERwin XML file.
4. If you have set up global mappings, select the Use Global Mappings check box. For details, see Managing global mappings (page 147).
5. Select one of the following Read entities from options:
   • Logical model - Allows you to import logical entities and attributes.
   • Physical model - Allows you to import physical entities and attributes, exactly as they appear in the source database.
6. Select a source database and then click OK.
   The Select Tables/Views window opens.
7. Continue from Step 4 in Generating the model (page 142).

Managing global mappings

Before you import a data model from ERwin, you can set up the global mappings from the logical ERwin model (the entities and attributes) to the physical source database (the tables and columns). This is useful if numerous entities in your model contain the same attribute. For example, lets assume that twenty source entities contain an attribute called "BusinessKey". In the physical source tables however, this column (which also appears in twenty tables) is called "Key". Using the Global Mappings feature, you only need to define the "Key-to-BusinessKey" mapping once instead of twenty different times.

When you import from ERwin, you can then select the Use Global Mappings check box to apply these mappings. See also Importing the model from ERwin (page 147).

You can add, edit, and remove entity and attribute mappings. If needed, you can also change the source database referenced for the tables (if you have several different sources defined).
To manage global mappings:

1. In the Model panel, from the drop-down menu in the top right, select Global Mappings. The Global Mappings window opens in the Tables to Entities tab.
2. Import the ERwin entities:
   1. Click Import Entities to Mappings toolbar button. The Import Entities window opens.
   2. In the File Path field, enter the full path to the ERwin.xml file (on the Compose Server machine) that includes the entities you want to import.
   3. Click OK.
3. Verify that Qlik Compose is using the desired source database. The database name is displayed in green at the bottom right of the toolbar.
   To select a different source database:
   1. Click Change Source Database.
   2. In the Set Source Database window, select a different database and then click OK.
4. Add new entities, edit existing entities, or remove entities as described in the following table:

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add a new entity</td>
<td>1. In the Tables to Entities tab, click the New toolbar button.</td>
</tr>
<tr>
<td></td>
<td>2. Next to the Entity Name field, click the browse button.</td>
</tr>
<tr>
<td></td>
<td>The Unmapped Entities window opens, listing only entities that have not</td>
</tr>
<tr>
<td></td>
<td>yet been mapped.</td>
</tr>
<tr>
<td></td>
<td>3. Select an entity and click OK.</td>
</tr>
<tr>
<td></td>
<td>4. Next to the Table Name field, click the magnifying glass icon.</td>
</tr>
<tr>
<td></td>
<td>The Find Table for [Entity Name] window opens for the selected entity.</td>
</tr>
<tr>
<td></td>
<td>5. From the Tables drop-down list on the left, select the table to map to.</td>
</tr>
<tr>
<td></td>
<td>6. Click OK. Qlik Compose populates the Table Schema field</td>
</tr>
<tr>
<td></td>
<td>automatically, based on the table you selected.</td>
</tr>
<tr>
<td></td>
<td>7. Repeat these steps for all unmapped entities.</td>
</tr>
<tr>
<td>Edit an entity</td>
<td>1. Move the mouse cursor over the entity and click the Edit button (pencil</td>
</tr>
<tr>
<td></td>
<td>icon) that appears on the right.</td>
</tr>
<tr>
<td></td>
<td>2. Make the required changes and click OK.</td>
</tr>
<tr>
<td>Delete an entity</td>
<td>1. Select the entity.</td>
</tr>
<tr>
<td></td>
<td>2. In the Entities toolbar, click Delete.</td>
</tr>
<tr>
<td></td>
<td>3. When prompted to confirm the deletion, click Yes.</td>
</tr>
<tr>
<td>Search for an entity</td>
<td>In the Search look-up field, start typing. Qlik Compose only displays</td>
</tr>
<tr>
<td></td>
<td>entities that match the search string.</td>
</tr>
</tbody>
</table>
5. Add, edit, or remove attributes as described in the table below:

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
</table>
| Add a new attribute | 1. In the **Columns to Attributes** tab, click the **New** toolbar button.  
                           2. Provide a name and description (optional) for the attribute and the column.  
                           3. Click **OK**.                                      |
| Search for an attribute | In the **Search** look-up field, start typing. Only attributes that match the search string will be displayed.  
                                *When searching for an attribute based on the attribute name, you must add the prefix "name: ". For example, if you want to search for an attribute that contains “ar” in its name, type "name: ar" in the **Search** look-up field.* |
| Edit an attribute    | 1. Move the mouse cursor over the attribute and click the **Edit** button (pencil icon) that appears on the right.  
                           2. Make the required changes and click **OK**.          |
| Remove an attribute  | 1. Select the attribute.  
                           2. Click the **Delete** toolbar button.  
                           3. When prompted to confirm the deletion, click **Yes**. |

6. Click **Close**.

**Model limitations**

- When Amazon Redshift is the data warehouse type, attribute names that contain the open parenthesis character "(" are not supported. If any of your attribute names contain the "(" character, you should remove it before creating the data warehouse tables. 
  *For information on renaming attribute names, see Add an attribute to all Satellite tables and the Hub table (page 157).*

- Discovering new tables does not affect existing entities in the model, even if there is a relationship between the new entity and one of the existing entities. For example, in the source database, Table 1 has a Foreign Key that points to Table 2. If Table 1 is added to the model and then Table 2 is added later, Table 1 will not be updated to contain the required Foreign Key.

- The data warehouse needs to be "adjusted" when deleting a relationship/attribute from the model and then adding the same relationship/attribute back to the model. However, the "Adjust" operation deletes the data from the corresponding data warehouse column.

**Validating the model**

Once you have generated the model, you can easily check that it is valid. For example, for a model to be valid, each of the tables must have a Business Key.
To validate the model:

1. Either click the **Validate** button in the bottom right of the **Model** panel.
   OR
   
   Select **Validate** from the drop-down menu in the top right of the **Model** panel.
   
   The **Validate Model** window opens.
   
   a. If the model is valid, a message will confirm the model’s validity. If the model is not valid, a list of invalid tables/views will be displayed.
   
   A message indicating why the entity is invalid will be displayed in the **Message** column.
   
   b. To resolve the issue, click the **Edit Entities** button to the right of the entity.
   
   The **Edit Model** window opens showing the invalid entity.

2. Resolve the issue (in this case, by adding a Business Key) and then click **Close**.
   
   A message will confirm the model’s validity.

3. Click **Close** to close the **Validate Model** window.

**Displaying the model**

Displaying the model is a good way to see the relationships between the various tables and/or views in your model.

**To display the model:**

Either click the **Display** button in the bottom right of the **Model** panel.

-OR-

Select **Display** from the drop-down menu in the top right of the **Model** panel.

The **Display Model** window opens showing the **Diagram** tab.
Diagram tab

In the **Diagram** tab, the following options are available:

- **Zoom** - Increase or decrease the magnification using the slider at the top right of the screen. Click the button to the right of the slider to restore the default size.
- **Search** - The ability to search for entities is particularly useful in a large model. To search for an entity, type a search string in the Search box. Compose lists the names of entities that match the search string. Select the desired entity.

- **Drag the diagram** - In addition to zooming, you can also drag the diagram by clicking the space around the diagram and dragging. This is useful for very large diagrams where zooming out would
render the text unreadable. The guide at the bottom right of the window shows you which part of the diagram is currently displayed.

- **Show/Hide all attributes for a selected entity** - Select an entity and then select/clear the **Attributes** check box in the top left of the window.
- **Show/Hide all business keys in the model** - Select/Clear the **Keys** check box in the top left of the window.
- **Show/Hide relationship attributes** - Right-click an entity and select this option to to show/hide the entity’s relationship attributes.
- **Show/Hide business keys** - Right-click an entity and select this option to to show/hide the entity’s business keys.
- **Change the Diagram Direction** - Select one of the available options from the **Direction** drop-down list at the top of the window.
- **Set as relationship source** - See *Creating and managing relationships* (page 161).
- **Hide this node** - Right-click an entity and select this option to show/hide the entity. To show the entity, click the **Hidden Nodes** box in the left of the window.
- **Hide selected nodes** - Right-click an entity and select this option to show/hide selected entities. To show the hidden entities, click the **Hidden Nodes** box in the left of the window.
- **Hide non-selected nodes** - Right-click an entity and select this option to show/hide non-selected entities. To show the hidden entities, click the **Hidden Nodes** box in the left of the window.
- **Invert selection** - Right-click an entity and select this option to highlight all entities except the selected entity.
- **Select all** - Right-click an entity and select this option to highlight all entities in the model.
- **Select path** - To highlight the path to which an entity belongs, either hover your mouse cursor over the entity or right-click the entity and select **Select Path**.
- **Select path and hide all other nodes** - Right-click an entity and select this option to highlight the entity’s neighbors.
- **Edit** - Either double-click the entity or right-click an entity and select the **Edit** option to edit the entity’s attributes.
- **Lineage** - Right-click an entity and select this option to show/hide the entity’s lineage. For more information on lineages, see *Lineage and impact analysis* (page 164).

**Tree View tab**

In the **Tree View** tab, the following options are available:

- **Search for an entity or attribute** - To search for a specific entity or attribute, enter a part of the name in the **Search** box. Entities that match the search string will be highlighted.
- **Expand/Collapse** - Click the arrow to the left of a table to see its attributes or related tables. To show or hide all sub-tables and table attributes, click the **Expand All/Collapse All** buttons at the top of the **Tree View** tab.
- **Lineage** - To see an entity or attribute’s lineage, hover your mouse over a table or attribute and then click the button that appears to its right.
For example, clicking the button next to the City attribute (shown in the image above) will open the following window:

For more information on lineages, see Lineage and impact analysis (page 164).

Managing the model

You can manage the model according to your needs, as described in the following topics:

- The Manage Model window (page 154)
- Managing entities (page 154)
- Creating and managing relationships (page 161)
- Managing attributes (page 155)
- Bulk Editing History types and Satellite numbers (page 164)
- Lineage and impact analysis (page 164)

There are two ways of editing a model in Compose:

- In the Manage Model window - Editing the model in the Manage Model window is preferable if you need to make several changes to the model as it provides access to all of the model’s entities and attributes. To display the results of your changes, open the Model Display window as described in Displaying the model (page 150).
- From the Model Display - Editing the model from the Model Display window is convenient if you only need to edit one or two entities. Another advantage of this method is that it allows you to see the result of your changes (in the entity relationship diagram) immediately.

To open the Manage Model window from the Model panel:

1. Click the Manage button at the bottom left of the Model panel or click the Entities link in the Model panel.
   The Manage Model window opens.
2. Edit the model according to the descriptions below.
To open the Manage Model window from the Model Display window:

1. Open the Model Display window as described in Displaying the model (page 150).
2. Double-click the entity you want to edit.
   The Manage Model window opens.
3. Edit the model according to the descriptions below.

The Manage Model window

The Manage Model window is split into two tabs: The Logical Model tab and the Physical Model tab. The Logical Model tab shows the entities and attributes as they appear in the model whereas the Physical Model tab provides a preview of the actual tables (and columns) that will be created in the data warehouse. So, for example, although the Categories table appears as a single entity in the Logical Model tab, it will appear as two tables (TDWH_Categories_HUB and TDWH_Categories_S01) in the Physical Model tab. The reason for this is because the logical Categories entity contains both Type 1 and Type 2 attributes. Type 1 attributes will be created as columns in the HUB table while Type 2 attributes will be created as columns in the Satellite table (S01). For more information on Type 1 and Type 2 attributes, see History.

All editing tasks are performed in the Logical Model tab, except for the following tasks which are performed in the Physical Model tab:

- Designate a Distribution Key Column (Amazon Redshift data warehouse only)
- Designate a Distribution Method (Microsoft Azure Synapse Analytics only)

For more information, see Defining Table Creation Modifiers (page 166).

Managing entities

You can add, edit and remove entities from your model as described in the table below.

<table>
<thead>
<tr>
<th>Entity management options</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>To</strong></td>
</tr>
<tr>
<td>Add an entity</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Edit an entity</td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>

All of the options available in the toolbar are also available from the drop-down menu in the toolbar. This is useful when you reduce the window size, since some of the toolbar buttons - or all of them depending on how small you make the window - will be hidden. The only button that will not be hidden regardless of the eventual window size is the drop-down menu button.
## Data Warehouse projects

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
<tbody>
<tr>
<td>Remove an entity</td>
<td>1. Click the <strong>Delete</strong> button in the <strong>Entities</strong> toolbar.</td>
</tr>
<tr>
<td></td>
<td>2. When prompted to confirm the deletion, click <strong>Yes</strong>.</td>
</tr>
<tr>
<td>Duplicate an entity</td>
<td>1. Select the entity you want to duplicate and then click the <strong>Duplicate</strong> button in the <strong>Entities</strong> toolbar.</td>
</tr>
<tr>
<td></td>
<td>2. Edit the entity’s name and description (optional) and then click <strong>OK</strong>.</td>
</tr>
<tr>
<td></td>
<td>The duplicated entity is added to the Entities list.</td>
</tr>
<tr>
<td>Import entities from</td>
<td>See <em>Importing entities and mappings from another project</em> (page 146).</td>
</tr>
<tr>
<td>another project</td>
<td></td>
</tr>
<tr>
<td>Import entities from</td>
<td>See <em>Importing the model from ERwin</em> (page 147).</td>
</tr>
<tr>
<td>ERwin</td>
<td></td>
</tr>
</tbody>
</table>

### Managing attributes

You can add, edit and remove attributes as required. All attributes in the model belong to the Attributes Domain. When adding a new attribute, you can either select an existing attribute from the Attributes Domain or create a new Attributes Domain. Both of these options are described in the table below.
## Attribute management options

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
</table>
| Add an attribute from the attributes domain | 1. Click the **New Attribute** button in the **Attributes** toolbar. The **New Attribute** window opens.  
2. To designate the attribute as a business key, select the **Key** check box.  
3. From the **Attribute domain** drop-down list, select the attribute domain you wish to add.  
4. To edit the selected attribute domain on-the-fly, click the edit button located after the **Attribute domain** drop-down list. This will open the **Edit - AttributeDomainName** window. Then, continue from Step 2 in [Edit an attribute domain](#).  
5. In the **Attribute name** field, optionally change the default instance name for the attribute domain. You can create multiple instances of a single Attribute Domain. This is especially useful if you want to use the same Attribute Domain across multiple tables, with each "instance" having its own unique name. This also allows you to edit the properties of each attribute without affecting the other attributes, despite all the Attribute Domain instances sharing a common Attribute Domain. For example, if the Attribute Domain name is "ID", you could create one instance for it in the "Categories" entity named "CategoryID" and another instance in the "Employees" entity named "EmployeeID". If, however, you edit the parent Attribute Domain attribute, all instances of that attribute will be updated as well.  
6. To add a prefix to the attribute name, enter the desired prefix in the **Prefix** field. Adding a prefix to an attribute name allows you to add multiple instances of the same attribute domain. For example, the attribute "Employee" could become two different attributes: "ReportsTo_Employee" and "HiredBy_Employee".  
7. Set the **History** Type and Satellite number. When the **History** Type is set to 2, a new record will be created in the data warehouse each time an attribute value changes.  
8. In the **Satellite/Hub** field, optionally change the satellite number. Note that the satellite number can only be changed when the **History Type** is set to 2. For an explanation of why this is so, see [The Manage Model window](#) (page 154).  
9. To add an expression, click the **fx** button located after the **Expression** field and then continue from [Creating expressions](#) (page 168).  
10. Click **OK** to save your settings. |
## To Do

<table>
<thead>
<tr>
<th>Task</th>
<th>Description</th>
</tr>
</thead>
</table>
| Create a new attribute domain and add it to the model | 1. Click the **New Attribute** button in the **Attributes** toolbar. The **New Attribute** window opens.  
2. To designate the attribute as a business key, select the **Key** check box.  
3. Click the plus sign to the right of the **Attribute domain** drop-down list. The **New Attribute Domain** window opens.  
   a. Specify a **Name** for the attributes domain.  
   b. From the **Type** drop-down list, select one of the available data types.  
   c. If the selected data type requires further configuration, additional fields will be displayed. For example, when Decimal is selected, the **Length** and **Scale** fields will be displayed. Set the values as desired.  
   d. Optionally, specify a **Description**.  
   e. Click **OK** to add the newly created attribute domain to the **Attribute domain** field and close the **New Attribute Domain** window.  
4. Continue from **Step 5 in Add an existing attribute domain** above. |

You can also add new attribute domains via the **Manage Attribute Domains** window. For more information, see Managing the **Attributes Domain** (page 160). |

<table>
<thead>
<tr>
<th>Task</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add a relationship</td>
<td>See Creating and managing relationships (page 161).</td>
</tr>
<tr>
<td>Add an attribute to all Satellite tables and the Hub table</td>
<td>You can use the <strong>Add to all Satellites and Hub</strong> option to define the same Primary Index for the Hub table and all Satellite tables. Select the desired attribute and then click the <strong>Add to all Satellites and Hub</strong> toolbar button. The attribute is added to the Hub table and to all the Satellite tables.</td>
</tr>
</tbody>
</table>
| Edit an attribute | **Method 1:**  
1. Select the attribute you want to edit and then click the **Edit** button in the **Attributes** toolbar. The **Edit - AttributeName** window opens  
2. Continue from **Step 2 of Add an attribute from the attributes domain** above.  
**Method 2:**  
1. Double-click the attribute you want to edit. The values in the attribute row become editable.  
2. Continue from **Step 2 of Add an attribute from the attributes domain** above. |
### To  |  Do This
--- | ---
Bulk edit history types and satellite numbers  |  See *Bulk Editing History types and Satellite numbers* (page 164).  
Show an attribute's lineage  |  See *Lineage and impact analysis* (page 164).  
Remove an attribute  |  1. Select the attribute(s) you want to delete.  
          2. Click the **Delete** button in the **Attributes** toolbar.  
          3. When prompted to confirm the deletion, click **Yes**.  
Change the attribute order  |  Select the attribute you want to move and use the **Move Up/Move to Top** and **Move Down /Move to Bottom** toolbar buttons to move the attribute.  
Search for an attribute  |  In the **Search** lookup field, start typing. Only attributes that match the search string will be displayed.  
Manage the Attributes Domain  |  See *Managing the Attributes Domain* (page 160).  
Create an expression for an attribute  |  See **Add an attribute from the attributes domain** or **Edit an attribute** above.  
Export the attributes to a CSV file  |  Select an entity from the Entities list on the left of the **Manage Model** window and then select **Export to CSV** from the drop-down menu in the Attributes toolbar. Depending on your browser settings, you will either be prompted to download the `<entityname>_Attributes.csv` file or it will be downloaded to your default Downloads location.  

#### Setting up derived attributes

Derived attributes are attributes whose data is "derived" from other attributes. For example, lets assume that the OrderDetails entity contains the attributes Quantity and UnitPrice but does not contain the attribute TotalPrice. To gain better insight into the annual sales figures, the organization would like to add the TotalPrice attribute to the model and derive its data from the Quantity and UnitPrice attributes.

Assuming that the Northwind sample database is the model's source, this could easily be done as follows:
1. Add the `TotalPrice` attribute domain to the model as described in *Managing attributes (page 155)*.
2. After finalizing the model, create the data warehouse tables as described in *Creating the data warehouse tables (page 179)*.
3. Click the `orderDetails` mapping as described in *Editing column mappings (page 189)*. Note that the `TotalPrice` attribute has no mapping as it was added after the Northwind source was discovered:

   ![Order Details Staging Columns](image)

4. Open the Expression Builder by clicking the `fx` icon to the right of the `TotalPrice` column name. Then, in the Expression Builder, add the `quantity` and `UnitPrice` columns to create the following expression:
   
   `Quantity*UnitPrice`

   For more information on creating expressions, see *Creating expressions (page 168)*.
5. Click **OK** to close the Expression Builder and save the expression. The `quantity` and `UnitPrice` landing zone columns are now mapped to the `TotalPrice` data warehouse column. Notice that the mapping lines are gray, indicating that the mapping is the result of an expression:

   ![Order Details Staging Columns](image)

   Hovering the mouse cursor over the gray lines highlights the derived column (`TotalPrice`) and the columns from which its data is derived (`Quantity` and `UnitPrice`).
Managing the Attributes Domain

The Attributes Domain provides a list of all the attributes available in the Compose model, as well as their data type. You can add, edit and delete attributes according to your data warehousing needs. The Attributes Domain also allows you to see which entities each attribute belongs to, as a single attribute may be present in several entities.

To manage the Attributes Domain

1. From the drop-down menu in the top right of the Model panel, select Attributes Domain.
2. Add, delete and edit attributes as describe in the table below.

### Attribute Domain management options

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
</table>
| Add an attributes domain | 1. Click the New Attributes Domain toolbar button. The New Attribute Domain window opens.  
2. In the Name field, specify a name for the attribute.  
3. From the Type drop-down list, select one of the available data types.  
4. If the selected data type requires further configuration, additional fields will be displayed. For example, when Decimal is selected, the Length and Scale fields will be displayed. Set the values as desired.  
5. Optionally specify a Description.  
6. Click OK to add the attribute and close the New Attribute Domain window. |

> Attribute domains names are case insensitive. For example, a project cannot contain one attribute domain called date and another called DATE.

| Edit an attribute domain | 1. Select the desired attribute and then click the Edit toolbar button. The Edit: Name window opens.  
2. Edit the attribute as described in steps 2-6 of Add an attributes domain above. Note that the Edit: Name window also contains a Used in Entities list. Knowing which entities the attribute is used in may affect the type of changes you make, as the planned changes may not be appropriate for all entities. |
### Creating and managing relationships

Similar to a foreign key, a relationship "attribute" is a special type of attribute that points to another entity in the same model. Typically, the relationship replaces the key attributes that connect an entity to a related entity. You can add, edit and delete relationships as required.

Possible reason for creating relationships are as follows:

- If your model is derived from the landing zone (as opposed to the source database(s)), the model will be created without any relationships
- Ensure data integrity between related entities

You can create relationship from the **Manage Model** window or from the **Display Model** window. Both of these methods are described below.

**When converting existing columns in a table with a relationship to another table, historical values may be lost and need to be loaded again or reinserted manually.**

### Adding relationships via the Manage Model window

1. Click the **Manage** button in the bottom left of the **Model** panel. The **Manage Model** window opens.
2. Select an Entity in the **Entities** list.
3. Click the **Add Relationship** button in the **Attributes** toolbar. The **Add Relationship From: Name** window opens.
4. From the **Add Relationship to Entity** drop-down list, select the entity to which you want to create a relationship.
5. If the originating entity contains attributes that were foreign keys in the source database, you can replace these attributes with Business Key attributes of the associated entity.
   **To do this:**
   a. Select the **Replace Existing Attribute(s) check box.** The left column shows the **Business Key Attributes of the Associated Entity.**
   b. From the **Attributes of Originating Entity** drop-down list on the right, select an attribute from the originating entity that was meant to be a foreign key.
6. If you want the relationship attribute to be a Business Key, select the **Business Key** check box. This option will only be displayed if the entity target can be designated as a Business Key.
7. Set the **History Type**.
8. Set a **Satellite Number**.


10. Optionally, enter a description.

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

### Adding relationships via the Display Model window

1. Click the **Display** button in the bottom left of the **Model** panel. The **Display Model** window opens.

2. Select one of the following methods:
   - **Method 1**: Right-click an entity and select **Add Relationship**. The **Add Relationship From: Name** window opens.
   - **Method 2**: Right-click an entity and select **Set as Relationship Source**. This method is useful if you need to search your model for the relationship target entity (since the source entity remains selected while you search).
   - **Method 3**: Select two entities by clicking them while holding down the [Ctrl] key. Then, right-click one of the entities and select the desired relationship from the context menu (according to the entity that you want to be the relationship source), as shown in the following example:

   ![Diagram showing relationships between entities]

3. If you selected **Method 2**, continue below. If you selected **Method 1**, continue from Step 4 in **Adding Relationships via the Manage Model window** above. If you selected **Method 3**, continue from Step 5 in **Adding Relationships via the Manage Model window** above.

4. Right-click the relationship target entity and select **Relationship Target for Relationship Source Name**. The **Add Relationship: Name** window opens with the relationship target entity already selected.

5. If the originating entity contains attributes that were foreign keys in the source database, you can replace these attributes with Business Key attributes of the associated entity.

   **To do this:**
   a. Select the **Replace Existing Attribute(s)** check box.
   b. The left column shows the **Business Key Attributes of the Associated Entity**.

---

Setup and User Guide - Qlik Compose, August 2021
b. From the **Attributes of Originating Entity** drop-down list on the right, select an attribute from the originating entity that was meant to be a foreign key.

6. If you want the relationship attribute to be a Business Key, select the **Business Key** check box. This option will only be displayed if the entity target can be designated as a Business Key.

7. Set the **History Type**.

> Since the history type for Business Keys must be type 1, the option to change the history type is unavailable when the **Business Key** check box is selected.

8. Set a **Satellite Number**.

> Since the satellite number for Business Keys must be "0", the option to change the satellite number is unavailable when the **Business Key** check box is selected.

10. Optionally, enter a description.
11. Click **OK** to save your settings.

**Preventing naming conflicts**

When a relationship from entity A to entity B is created, Compose implicitly adds entity B’s primary key columns to table A. This means that if there are two or more relationships from entity A to entity B, a column naming conflict will arise (as entity B’s primary key columns will be added to table A multiple times). Such conflicts can easily be avoided by adding a meaningful prefix to the relationship attributes in entity A, which will result in the prefix being added to the physical columns as well.

**Example:**

The **Orders** entity contains two attributes that are related to the **People** entity: the **Customer** and **Seller** attributes. Therefore, Mike wants to create two relationships from the **Orders** entity to the **People** entity. The primary key of the **People** table consists of the **FirstName** and **LastName** attributes. As there are two relationships, the primary key columns of the **People** entity will be added twice to the **Orders** entity. To prevent duplication errors, Mike adds the **Customer_** and **Seller_** prefixes to the relationship attributes in the **Orders** entity, which results in the physical columns **Customer_FirstName**, **Seller_FirstName**, **Customer_LastName**, and **Seller_LastName**.

**Deleting relationships**

1. Click the **Manage** button in the bottom left of the **Model** panel. The **Manage Model** window opens.
2. Select the relationship attribute you want to delete.
3. Click the **Delete** button in the Attributes toolbar. The **Delete Relationship** window opens.
4. To restore an attribute that was replaced when the relationship was created, select the **Restore original attribute(s)** check box. For more information about replacing attributes, see Step 5 in
Adding relationships via the Manage Model window above.
5. Click Yes to delete the relationship attribute.

Bulk Editing History types and Satellite numbers
Use the Bulk Edit feature to edit the History type and Satellite number of multiple attributes.

To bulk edit history types and satellite numbers:
1. Select the attributes whose History type and/or Satellite number you want to change and click the Bulk Edit toolbar button.
2. In the Bulk Edit window, change the History type and/or Satellite number as required.
3. Click OK close the Bulk Edit window and save your settings.

Lineage and impact analysis
Before editing an entity or attribute, you may want to see which other entities/attributes in the entity's/attribute's lineage will be impacted by the change. For example, removing the "Discount" attribute from a table will affect the "Total Price". Additionally, a single attribute may have different names depending on its location.

Places where you can view lineage in Compose:
- The Manage Model window described below.
- The Display Model window described in Displaying the model (page 150).
- When editing a data mart. For more information, see Managing data marts (page 219).

Top-level entities in the data mart fact will not be shown in the lineage. For example, if both the Orders and Order Details entities are used in a Fact, the Model lineage for Orders will show Order Details but not Orders.

To view the lineage of an entity or attribute:
1. Click the Manage button in the bottom left of the Model panel. The Manage Model window opens.
2. Display the lineage as described below:

Lineage procedures

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
<tbody>
<tr>
<td>Show an entity's lineage</td>
<td>Select the entity and select <strong>Show Lineage</strong> from the drop-down menu in the Entity toolbar.</td>
</tr>
<tr>
<td>Show an attribute's lineage</td>
<td>Select the attribute and click the <strong>Show Lineage</strong> button in the Attribute toolbar.</td>
</tr>
</tbody>
</table>
Adding Date and Time entities to your model

Compose provides built-in Date and Time entities that you can add to your model. This facilitates access to all attributes of date and time (such as day of the week, quarter, and so on) both in the BI reports and when creating transformations in the data mart.

The Date entity contains a record for every day. Dates in the Date entity range from January 1st 1900 to December 31st 2099.

The Time entity contains all the hours and minutes in a 24 hour period. When you create the data warehouse tables, the Date and Time entities are automatically populated with relevant data. You can view this data as described in Viewing the data warehouse tables (page 180).

Both the date and the time values are presented in multiple formats (e.g. 12 hour format or 24 hour format), allowing you to choose which format will be displayed in your BI reports. Other format include abbreviated forms of date and time, different month/year/day formats (e.g. 12/31/2017 as opposed to 2017-12-31), and so on.

You can either add the entities to a new project (before you create the Data Warehouse tables) or to an existing project. If you add them to an existing project's model, you will also need to validate and adjust the Data Warehouse as described in Validating the data warehouse (page 208).

You can even add custom date and time attributes to the entities from the tables in your landing zone. For example, if one of your source tables lists all the working days and non-working days, you can add an "Is Working Day" attribute to the Date entity and then load it from the relevant source table. Just like regular entities, Compose knows how to merge the incoming data of working and non-working days into the existing Date entity.

For an explanation of how to add attributes to an entity, see Managing attributes (page 155).

You cannot add relationships to the Date and Time entities. However, every date and time attribute has an implicit relationship to the Date and Time dimensions, which allows you to select the relevant dimension when creating your star schema in the data mart.

For information on working with Date and Time dimensions in the data mart, see Creating and managing data marts (page 211).

For all of the supported data sources except Oracle, you can add both Date and Time entities to your model. If you are using Oracle as your data source, you can only add the Date entity to your model. This is because Oracle does not have a data type specifically for Time.

To add Date and Time entities to your model:

1. Open the desired Compose project.
2. From the drop-down menu in the top right of the Model panel, select Add Date and Time entities or Add Date Entity if you data source is Oracle (see Note above).
3. When prompted to confirm the action, click Yes.
4. The Date and Time entities will be added to your model. By default, the Date and Time entities are hidden from the model display (as they are not related to other entities in your model). If you want to show them anyway, select the **Date and Time** model check box in the Data Warehouse Model window.

5. For information about displaying the model, see *Displaying the model (page 150).*

You can also delete the Date and/or Time entities if you no longer require them and add them again later.

**Defining Table Creation Modifiers**

You can set table modifiers for individual entities in the **Physical Model** tab, thereby overriding the default settings in the project settings' **Table creation modifiers tab (page 43)** Table modifiers allow you to append additional table properties to the default Compose CREATE TABLE statement.

The available options are located below the **Columns** list on the right of the tab, and are as follows:

- **Project settings default** - When this option is selected (the default), the settings from the project settings' **Table creation modifiers tab (page 43)** will be used.
- **Custom** - This option is useful for appending additional table properties to the default Compose CREATE TABLE statement. Leveraging this option requires SQL coding knowledge.
- **Custom distribution keys** - This option is useful if you only need to define custom distribution keys for individual entities. Although this can also be done using the **Custom** option (see below), the **Custom distribution keys** option is more convenient as it does not require any prior SQL knowledge.

- **Supported with Microsoft Azure Synapse Analytics and Amazon Redshift only.**
- **The default distribution key for all data warehouse tables is the ID column.**

**Setting table creation modifiers**

By default, Compose creates tables in the data warehouse using the standard CREATE TABLE statement. However, organizations often need tables to be created with custom properties for better performance, special permissions, custom collation, and so on. For example, in Microsoft Azure Synapse Analytics, it's possible to create a table as a HEAP, which is optimized for smaller tables. By default, Compose creates tables in Microsoft Azure Synapse Analytics as a CLUSTERED COLUMNSTORE INDEX, which offers the best overall query performance for large tables.

The procedure for setting table modifiers is as follows:

1. In the **Physical Model** tab, select the desired entity.
2. Select the **Custom** option.
3. Click the **Edit** button to open the Table Creation Modifier editor.
4. Enter the SQL parts you wish to append to the CREATE TABLE statement.
5. Optionally, but strongly recommended, validate the SQL in an external validation tool that supports your specific database and version. For instance, if you are validating
5  Data Warehouse projects

Compose does not provide any way of validating your SQL. Therefore, make sure to validate the SQL before deploying in a production environment.

6. Click **OK** to close the editor and save your SQL parts.

**Example of a Valid Table Creation Modifier**

In the following example, the Compose CREATE TABLE statement (rows 1-5) is appended with an SQL part instructing Compose to create the table as a HEAP (row 6).

```sql
CREATE TABLE MyTable
(
  column1 integer,
  column2 varchar(50),
)
WITH (HEAP)
```

**Setting Custom Distribution Keys**

This section describes how to set a custom distribution key for tables created in Amazon Redshift and Microsoft Azure Synapse Analytics. Note that depending on the selected **Distribution Style** (Amazon Redshift) or **Distribution Method** (Microsoft Azure Synapse Analytics), some of the options may not be available.

**Setting a distribution key for Amazon Redshift Data Warehouse**

Select and entity and then set a distribution key for Amazon Redshift Data Warehouse according to the table below.

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
</table>
| Set a distribution style  | From the **Distribution Style** drop-down, select **Even**, **Key** or **All**.  
|                           | For more information on distribution styles, see:  
|                           | [Distribution styles - Amazon Redshift](#) |
| Add a distribution key    | 1. Click the **Add Distribution Key** button.  
|                           | A row is added to the table displaying a drop-down list.  
|                           | 2. Select one of the available columns. |
| Edit a distribution key   | 1. Double-click the row.  
|                           | A drop-down list will be shown in the **Column** column.  
|                           | 2. Select one of the available columns. |
| Delete a distribution key | Select the distribution key and then click the **Delete** button. The key is deleted. |
| Change the position of a distribution key | Select the distribution key and then click the "Up" or "Down" buttons to move the key to the desired position. |
Setting a distribution key for Microsoft Azure Synapse Analytics

Select and entity and then set a distribution key for Microsoft Azure Synapse Analytics according to the table below.

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set a distribution method</td>
<td>From the Distribution Method drop-down, select Hash, Round Robin or Replicate.</td>
</tr>
<tr>
<td></td>
<td>For more information on the distributions options, see:</td>
</tr>
<tr>
<td></td>
<td>Guidance for designing distributed tables in Synapse SQL pool - Microsoft Azure</td>
</tr>
<tr>
<td>Add a distribution key</td>
<td>1. Click the Add Distribution Key button.</td>
</tr>
<tr>
<td></td>
<td>A row is added to the table displaying a drop-down list.</td>
</tr>
<tr>
<td></td>
<td>2. Select one of the available columns.</td>
</tr>
<tr>
<td>Edit a distribution key</td>
<td>1. Double-click the row.</td>
</tr>
<tr>
<td></td>
<td>A drop-down list will be shown in the Column column.</td>
</tr>
<tr>
<td></td>
<td>2. Select one of the available columns.</td>
</tr>
<tr>
<td>Delete a distribution key</td>
<td>Select the distribution key and then click the Delete button. The key is deleted.</td>
</tr>
<tr>
<td>Change the position of a distribution key</td>
<td>Select the distribution key and then click the &quot;Up&quot; or &quot;Down&quot; buttons to move the key to the desired position.</td>
</tr>
</tbody>
</table>

Creating expressions

Compose allows you to create data transformations in several different places according to your needs. A transformation can either be a filter (i.e. excluding certain data) or an expression (i.e. manipulating a single record). The table below lists the places where transformations can be created and provides reasons for creating the transformation in each of the specified places.

Changes in a dimension expression or lookup of a column in a dimension are not updated retroactively. In order to update historical data, you would need to reload the data which could take a long time depending on the number of records and their history.
### Data transformation location comparison

<table>
<thead>
<tr>
<th>Where the Transformation is Created</th>
<th>Reasons to Create a Transformation There</th>
<th>When the Transformation is Applied</th>
</tr>
</thead>
</table>
| Replicate                           | • Filtering large amounts of data that is not needed for the data warehouse (in the present or the future)  
• Obfuscation due to regulatory reasons or internal policies  
• Data type conversion (e.g. converting a source data type that is not supported on the data warehouse platform) | Before the data reaches the landing zone. |
| Model                               | • The default location if you are not sure where to put it  
• General business logic  
• Needed for several sources or several data marts | Applied as an update to the staging tables after creating the mappings. |
| Data Warehouse                      | • Specific source preparation  
• Needed for merging several sources | Between the landing zone and the staging zone. |
| Data Mart                           | • Specific to a data mart  
• Managed by a data mart data specialist | Between the data warehouse and the data mart. |

See also *Defining reusable transformations (page 176).*

The following topics describe the Expression Builder:

- *Opening the expression builder (page 169)*
- *Expression builder overview (page 170)*
- *Building expressions (page 171)*
- *Testing expressions (page 172)*

### Opening the expression builder

The Expression Builder enables you to create a transformation without needing to type anything manually.

The Expression Builder can be opened in several places, depending on your needs. For more information about where to create a transformation, see the table in *Creating expressions (page 168).*
Expression builder overview

The following section provides an overview of the Expression Builder functionality.

The Expression Builder consists of the following panels:

- **Tabs on the left of the Expression Builder**: These tabs contain elements that you can add to an expression. Select elements and add them to the **Build Expression** pane to create an expression. For more information, see *Building expressions* (page 171).

  The following tabs are available:

  - **Parameters** - Only displayed when opening the Expression Builder from within the **Reusable Transformations > Edit Transformation** window. For information on reusable transformations, see *Defining reusable transformations* (page 176) below.
  - **Input Columns/Input Attributes** - Columns/attributes that can be used to build your expression.
  - **Transformations** - Contains a list of reusable transformations. The tab is not displayed if no reusable transformations have been defined. For information on reusable transformations, see *Defining reusable transformations* (page 176) below.
  - **Operators** - Operators that can be used to build your expression.
  - **Functions** - Functions that can be used to build your expression.
The Operators and Functions displayed in the Expression Builder use SQL format. As SQL support and implementation is different for each data warehouse (i.e. database) type and version, the data warehouse being used in your Compose project will determine which Operators and Functions will be available. For example, functions introduced with Microsoft SQL Server 2017 will not work if the database being used for the data warehouse is Microsoft SQL Server 2015.

Additionally, the list of Operators and Functions displayed in the Expression Builder is not comprehensive. However, you can use any Operators and Functions supported by the data warehouse, even if they are not included in the list.

For an explanation of the available Operators and Functions, refer to the Help for your data warehouse.

- **Build Expression Pane:** The Build Expression pane is where you build your expression. You can add elements, such as columns or operators to the panel as well as type all or part of the expression. For more information, see Building expressions (page 171).

- **Parse Expression Pane:** This pane displays the parameters for the expression. After you build the expression, click Parse Parameters to list the expression parameters. You can then edit the parameters, enter a value for each of the parameters and associate attributes with them. For more information, see Parsing expressions (page 172).

- **Test Expression Pane:** 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 Testing expressions (page 172).

Building expressions

The first step in using the Expression Builder is to build an expression in the Build Expression pane.

To add operators to your expression, you can use the Operator tab on the left or the Operator buttons located above the Build Expression pane or any combination of these.

To build an expression:

1. Hover the mouse cursor over the element that you want to add to your expression (expressions usually start with an Input Column) and click the arrow that appears to its right.
2. Add Operators additional Input Columns and Functions as required.

To add operators to your expression, you can use the Operator tab on the left or the Operator buttons located above the Build Expression pane or any combination of these.
Example:

To create an expression that combines the FirstName name and LastName columns, do the following:

1. Add the FirstName Input Column to the Build Expression pane.
2. Assuming that Microsoft SQL Server is the data warehouse, in the Operator toolbar above the Build Expression pane, click the concatenate (+) operator.
3. Then add a space between single quote characters and click the concatenate (+) operator again.
4. Add the LastName Input Column to the Build Expression pane.

The expression would look like this:

```
${FirstName}+' '+${LastName}
```

Parsing expressions
When you add operators to the expression, the expression’s parameters are usually added automatically to the Parse Expression pane. However, when you complete your expression or edit it, you may need to parse the expression see all of the parameters.

To parse the expression parameters:

- Click the Parse Expression button below the Build Expression pane.

If the expression is not valid, a red error message will appear at the bottom of the Expression Builder window.

If the expression is valid, the expression parameters and attributes (Input Columns) will be displayed in the in the Parse Expression pane. See Testing an expression (page 174).

Editing parameter names
By default, the parameter name is the same as the input column name. However, you can change the parameter name as needed and then associate it with an input column. This is useful, for instance, when you need to shorten attribute names. For example, EstimatedTimeOfArrival can be abbreviated to ETA.

To edit a parameter and associate it with an input column:

1. In the Parse Expression pane, edit the parameter name as required.
2. From the Attribute drop-down list, select the desired input column.

Testing expressions
You test your expression to check that results are as expected. The following figure is an example of an expression that has been evaluated and tested.
Certain expressions may fail during runtime, even though clicking **Test Expression** in the Expression Builder indicated that they were valid.

This is because clicking **Test Expression** runs a query whereas during runtime, the expression is run as a sub-query. This issue arises partly because the rules that govern queries are slightly different from the rules that govern sub-queries.

For example, a semi-colon (;) is allowed in a query but not in a sub-query.

Testing an expression that contains an analytic function will validate the syntax without actually executing the function. Additionally, the test will only be performed on a single record.

Compose does not check the data types of columns used in an expression for compatibility. For example, if a column of type integer is used in an expression for a column of type varchar, the expression will not be executed successfully.
Testing an expression

1. In the Expression Builder window, build an expression as described in Building expressions (page 171).
2. Click Parse Expression as described in Parsing expressions (page 172).
3. View the parameters that are displayed. If your expression is not valid, an error message is displayed.
4. Optionally edit the parameters name(s) as described in Editing parameter names (page 172).
5. Type values for each parameter and then click Test Expression to see the expression result. For example, using the expression in Testing an expression (page 174), type Mike for FirstName and Smith for LastName. The result displayed is Mike Smith.

To test an expression:

1. In the Expression Builder window, build an expression as described in Building expressions (page 171).
2. Click Parse Expression as described in Parsing expressions (page 172).
3. View the parameters that are displayed. If your expression is not valid, an error message is displayed.
4. Optionally edit the parameters name(s) as described in Editing parameter names (page 172).
5. Type values for each parameter and then click Test Expression to see the expression result. For example, using the expression in Testing an expression (page 174), type Mike for FirstName and Smith for LastName. The result displayed is Mike Smith.
6. This step is only available for transformations created in the **Edit Mappings** window. When you create a transformation in the **Edit Mappings** window, an additional button called **Show Data** appears to the left of the **Test Expression** button. You can click this button to see how your expression translates into actual data.

   For example, clicking the **Show Data** button for the expression \( \text{UnitPrice} \times \text{Quantity} \) will open the following window.

![Order Details](image)

For more information on the **Edit Mappings** window, see **Editing column mappings** (page 189) in **Creating and managing the data warehouse** (page 177).
Defining reusable transformations

In a single Compose project there may be several processes that require similar data transformations. For example a reusable transformation can be defined that concatenates first and last names. This transformation could then be used both in the Customers mapping and in the Employees mapping.

As opposed to stored functions or procedures which are environment dependent, reusable transformations are environment agnostic, meaning that not only can they be used as required within a Compose project, but they can also be used across different environments (using Compose’s export/import function).

Centrally managed transformations increase efficiency by eliminating unnecessary duplication, while at the same time, enabling the seamless propagation of changes to all transformation instances.

To define a reusable transformation:

1. From the drop-down menu in the top right of the Model panel, select Reusable Transformations. The Reusable Transformations window opens. The window is split into the following panes:
   - Upper pane - Lists the reusable transformations that have been defined.
   - Lower pane - Provides additional information about transformation instances such as where they are in use (e.g. mappings, model, etc.) and the expression that was created using the transformation.

   Select a transformation to see the additional information.

   1. In the Name field, specify a name for the transformation.
   2. In the Category field, specify a category name. If the category name already exists it will be displayed below the field when you start to type the name. To group the new transformation in the same category, simply select the existing name (unless of course you wish to create a new category with a similar name).

   In the Expression Builder, transformations are grouped according to their category name, making it easier to find the transformation you want to use. Therefore, when specifying a category name, it is recommended to choose a name that reflects the purpose of the transformation. For example, if you create several transformations that concatenate data, it would make sense to group those transformations under a category called "Join".

3. To add a parameter to the transformation, click the New button to the right of the Parameters heading.
   A new row is added to the Parameters list.

4. Specify a name for the parameter, select an appropriate data type, and optionally provide a description.

   *If you add multiple parameters, you can change a parameter’s position by selecting the parameter and then using the Up/Down arrows (above the Parameters list) to reposition it.*

5. Click the Create Expression button below the Parameters list.
The **Edit Transformation** window opens.

6. In the **Edit Transformation** window, create an expression using the parameters you defined earlier.

   For information on creating expressions, see *Creating expressions (page 168).*

7. Click **OK** to save the transformation.

   The transformation is added to the list in the upper pane.

Once a transformation has been defined, it will be available for selection as needed in the Expression Builder’s **Transformations** tab.

For information on creating expressions, see *Creating expressions (page 168).*

**Managing reusable transformations**

You can manage reusable transformation as described in the table below.

<table>
<thead>
<tr>
<th>Reusable transformation management options</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>To</strong></td>
</tr>
<tr>
<td>Delete a transformation</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Edit a transformation</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Edit a parameter</td>
</tr>
</tbody>
</table>

**5.7 Creating and managing the data warehouse**

Once your model is set up properly, the next step in the Compose workflow is to create the data warehouse tables, generate the task(s) and run the data warehouse task.

In this section:

- *Data warehouse tasks (page 178)*
- *Managing tasks (page 187)*
- *Viewing and exporting task statements (page 202)*
- *Modifying task settings (page 203)*
5 Data Warehouse projects

- Validating the data warehouse (page 208)
- Clearing the data warehouse metadata cache (page 210)

Data warehouse tasks

This section describes how to create the data warehouse tables, generate the task and run a data warehouse task. It contains the following topics:

- How Compose handles missing references in the data warehouse (page 178)
- Creating the data warehouse tables (page 179)
- Generating the task (page 182)
- Controlling data warehouse tasks (page 183)

How Compose handles missing references in the data warehouse

Before running a data warehouse task, it is important to understand how Compose handles missing references. Missing references may be involve records that are simply missing or records whose arrival has been delayed. The latter might occur if data is ingested from two different systems (for example, an ERP system and a CRM system), with each system having its own task.

Handling an early-arriving fact:

If a record references another record which does not exist yet, then Compose will do the following:

- Insert a placeholder for the missing reference record. The placeholder record will only include the business key and surrogate key. The rest of the columns will be set to NULL.

  The fact being processed can already include a valid reference to the surrogate key of the reference record.

- Document the missing record in the TLOG_REF_ERRORS_VALUES table. The TLOG_REF_ERRORS_VALUES table contains the following columns:
  - RUNNO - The task run number.
  - RELATIONNR - An internal number that can be used by Qlik Support to determine the source entity.
  - NO_RELATIONS - The number of missing references. For example, if Customer A ordered three different items (from the Orders table) and Customer A is missing, this number will be three.
  - KEYVALUE1-20 - The missing record. Since the missing record is a Primary Key, which may consist of several columns, there are 20 KEYVALUE columns.

Example:

If the "Orders" table references "SuperGlue" in the "Products" table, but "SuperGlue" does not exist in that table, Compose will mark "SuperGlue" as a missing reference, insert a record with the key value "SuperGlue" (assuming that the product name is the business key) to the "Products" table, and insert NULL values in the remaining "Products" table columns.
When the missing reference eventually arrives, it will be mapped to the record created for it and the NULL values will be replaced by the actual values.

> If the record is defined as history type 2, the record with the NULL values will remain as a historical record.

See also: Viewing missing references (page 242).

Creating the data warehouse tables

Compose creates two types of data warehouse tables: staging tables (indicated by the TSTG prefix) and the actual data warehouse tables (indicated by the TDWH prefix).

In addition, Compose automatically creates views for the TDWH tables in the following format:

<schema_name>.VDWH.<entity_name>[satellite_number_if_several]

**Example:**

dbo.VDWH_Customers02

For each entity, Compose creates a single view containing both the satellite data and the associated hub data (or only the hub data if the entity has no satellites). If an entity has several satellites, then Compose will create a view for each of the satellite tables. In such a case, the view name will be suffixed with the user-defined satellite name as in the example above.

Compose for Data Warehouses adds RUNNO_INSERT and RUNNO_UPDATE columns to both the Data Warehouse tables and the data mart tables. These columns contain the ETL task run number, which can be used in the Run Details window or in the Details tab to find out more information about the task (e.g. the number of rows updated or inserted per table). Note that in hub tables and type 1 dimensions, the RUNNO_UPDATE number will usually be higher than the RUNNO_INSERT number as these tables do not contain any history. In satellite tables or type 2 dimension tables however, the RUNNO_INSERT number and the RUNNO_UPDATE number will always be the same as a new row is inserted for each update (i.e. history is retained).

> Data Warehouse views that contain both hub and satellite data will contain two RUNNO_INSERT and two RUNNO_UPDATE columns. The hub table RUNNO columns are appended with an ".H" (e.g. RUNNO_INSERT_H) while the satellite table table RUNNO columns are appended with an ".S" e.g. RUNNO_UPDATE_S.

To create the data warehouse tables:

1. Click the Create button in the bottom right of the Data Warehouse panel. The Creating Data Warehouse window opens.
   A progress bar indicates the current progress. For each stage of the Data Warehouse generation process, a corresponding message appears in the Messages list.
When creating table in a Microsoft SQL Server data warehouse, you may encounter the following error:

Data warehouse creation failed. Error: Cannot create a row of size 11272 which is greater than the allowable maximum row size of 8060.
The statement has been terminated.
This is a well-documented Microsoft SQL Server limitation. To work around this limitation you need to split the offending table(s) into smaller tables.

2. When the "Data warehouse created successfully" message appears, click Close.

Viewing the data warehouse tables

After the data warehouse tables are created, you can view them by clicking the number to the left of the Data Warehouse Tables Present text in the Data Warehouse panel.

When you click the link, the Data Warehouse Tables window opens showing a list of all the tables in your data warehouse.

Compose for Data Warehouses adds RUNNO_INSERT and RUNNO_UPDATE columns to both the Data Warehouse tables and the data mart tables. These columns contains the ETL task run number, which can be used (in the Run Details window or in the Details tab) to find out more information about the task (e.g. the number of rows updated or inserted per table). Note that in hub tables and type 1 dimensions, the
**RUNNO_UPDATE** number will usually be higher than the **RUNNO_INSERT** number as these tables do not contain any history. In satellite tables or type 2 dimension tables however, the **RUNNO_INSERT** number and the **RUNNO_UPDATE** number will always be the same as a new row is inserted for each update (i.e. history is retained).

**Data Warehouse views that contain both hub and satellite data will contain two **RUNNO_INSERT** and two **RUNNO_UPDATE** columns. The hub table **RUNNO** columns are appended with an "_H" (e.g. **RUNNO_INSERT_H**) while the satellite table table **RUNNO** columns are appended with an "_S" e.g. **RUNNO_UPDATE_S**).**

**In the data mart tables, the **RUNNO_INSERT/RUNNO_UPDATE** column names are prefixed by the table name e.g. **ORDERS_RUNNO_UPDATE**.**

To view a specific table, simply double-click the table.

**Apart from the **Date** and **Time** tables which are automatically populated on creation, the other tables will be empty until you run the data warehouse task.**

See Controlling data warehouse tasks (page 183) below for information on running a data warehouse task.

In the **<Table Name>** window, you can perform the following tasks:

- Choose how many rows to display from the **Rows** drop-down list.
- Click the **Column Settings** button to choose to choose which columns will be displayed and the order in which they will be displayed.

**Creating a Change Processing task**

You can create a CDC task by duplicating the Full Load task. This is useful if you currently have a Full Load task only and the need (which wasn't previously anticipated) arises to capture changes from all of the tables or only from certain tables.

**To do this:**

1. Select the Full Load task in the left pane of the Manage Data Warehouse Tasks window and click the **Duplicate** toolbar button.
2. In the **Duplicate <ETL_Set_Name>** window, set the following properties:
   - **Task name** - The name for the task (e.g. employees_changes).
   - **Select a landing zone** - Select the same landing zone as the Full Load task.
   - **Schemas** - Select the same schema as the Full Load task.
   - **Task type** - Select **Change Tables**.
3. To only apply changes to selected tables, in the Mappings column, select the tables whose changes you want to apply to the data warehouse.

**Important:**
- If you want to modify a column’s mapping, the table name displayed in the left pane of the Edit Mappings window will not be appended with the "_ct" suffix (which is the default naming format for Replicate Change Tables). However, the changes will still be taken from the associated Change Table.
- For each Primary Key there is one record. Changing a Primary Key on the source record will cause a new record to be inserted in the data warehouse.

4. Click Generate and wait for the task statements to be generated.
5. To capture the changes and apply them to the data warehouse, click Run.

### Reloading data from the source tables to the Landing Zone

In cases of inconsistencies or when metadata from the source tables is not replicated to the Landing Zone, you may need to reload the data.

**To reload data from the source tables:**

1. Run the Qlik Replicate Full Load replication task again.
2. Run the Compose for Data Warehouses Full Load storage task.
3. Continue running the existing Compose Change Processing task.

**Important:**
- No duplicates are created, as Compose compares the records before adding them to the Data Warehouse.
- Records that should be deleted in the new Full Load task are not deleted nor marked for deletion (soft delete).

### Generating the task

After the data warehouse tables have been created, you then need to generate the task that will be used in the data warehouse task. The task contains the Mappings ETL (which is automatically created) and any custom ETLs that you have defined. If you need to make changes to the Mappings or define custom ETLs, continue from Managing tasks (page 187) and Creating custom ETL Scripts (page 184) respectively.

**To generate the data warehouse task:**

1. Click the Manage button in the bottom left of the Data Warehouse panel. The Manage Data Warehouse Tasks window opens.
2. If you have more than one task, in the left pane, select the task that you want to generate.
3. Do one of the following:
5 Data Warehouse projects

- To generate the task with all validations, click the Generate toolbar button.
- To generate the task with basic validations, click the inverted triangle to the right of the Generate button and select Basic validations from the drop-down menu.

By default, Compose generates the task with all validations. However, when many expressions and lookups are defined, generating with all validations may take a long time. In such a case, you can try generating with basic validations instead. With basic validations, validations that may take a while (such as those that access databases to verify the existence of columns used in expressions and lookups), will be skipped.

The Generating task for <Name> progress window opens. When the "Generate task finished successfully" message is displayed, close the window.

Only mappings selected in the Manage Data Warehouse Tasks window will be generated.

Controlling data warehouse tasks

Once the data warehouse tables have been created and the task has been generated, you can then proceed to run the data warehouse task. The data warehouse task extracts data from the landing tables, loads it into the staging tables, and finally loads the data into the data warehouse tables.

Ingesting a historical record deletes any history that is later than the ingested record. For example, if a data warehouse contains the following historical records:

2012 - Boston
2014 - Chicago
2015 - New Jersey

Ingesting the record 2013 - New York will delete the 2014 and 2015 records.

Data warehouse tasks can be run manually, scheduled to run periodically or run as part of a workflow. The section below describes how to run a data warehouse task manually. For information on scheduling data warehouse tasks or including them in a workflow, see Controlling and monitoring tasks and workflows (page 240).

Data warehouse tasks cannot run in parallel with data mart tasks. Data warehouse tasks that update the same tables cannot run in parallel.

To run a data warehouse task:

1. Click the Manage button in the bottom right of the Data Warehouse panel. The Manage Data Warehouse Tasks window opens.
2. If you have more than one task, in the left pane, select the task that you want to generate.
3. Click the Run toolbar button. The window switches to Monitor view and a progress bar shows the current progress in terms of percentage.
You can stop the task at any time by clicking the Abort toolbar button. This may be necessary if you need to urgently edit the task settings due to some unforeseen development. After editing the task settings, simply click the Run button again to restart the task.

**Aborting a task may leave the data warehouse tables in an inconsistent state. Consistency will be restored the next time the task is run.**

4. When the progress reaches 100% completed, close the Manage Data Warehouse Tasks window.

Other monitoring information such as the task details (i.e. the number of rows inserted/updated) and the task log files can be accessed by clicking the Run Details and Log buttons respectively.

Once the data warehouse has been successfully loaded into the data warehouse tables, you can then proceed to the final part of the Compose workflow - defining and populating data marts. For more information, see Creating and managing data marts (page 211).

### Creating custom ETL Scripts

In addition to the Mappings ETL, you can define custom ETLs as required. User-defined ETLs can perform a number of useful operations such as defining specific transformations, gathering statistics, performing cleansing, and filtering data.

**Common Table Expressions (CTEs) are not supported as well as some special clauses.**

#### To create a custom ETL:

1. Click the Manage button in the bottom left of the Data Warehouse panel. The Manage Data Warehouse Tasks window opens.
2. Select one of the following tabs according to your needs:
   - **Pre Loading ETL** - to define an ETL that will manipulate the data before it is loaded from the landing tables to the data warehouse staging tables. When enabled, the Pre-loading ETL will be run even if there are no mappings or Replicate-generated source data associated with it, which is particularly useful for customer wanting to perform transformations on data generated by third-party tools.
   - **Multi Table ETL** - to define an ETL for multiple tables.
   - **Single Table ETL** - to define an ETL for a single table.
   - **Post Loading ETL** - to define an ETL that will be executed after the data has been loaded from the staging tables to the data warehouse.
3. If you selected **Single Table ETL**, select an entity in the Entity column and then click the New button above the Entity list. For **Multi Table** and **Post Loading** ETLs, just click the New button.
4. Specify a name for your ETL and then click OK.
   - If you selected **Single Table ETL**, the ETL is added as a link to the User Defined ETL column. If you selected **Multi Table ETL** or **Post Loading ETL**, the ETL is added as a link in their respective tabs.
5. Click the link to open the Edit ETL Instructions window.
6. If you selected Single Table ETL, select a column and click the arrow to the right of the selected column to add it to the ETL.
   If you selected Multi Table ETL or Post Loading ETL, select a table and a column and then click the arrow to the right of the selected table/column to add it to the ETL. Repeat as necessary.
7. Use the Select, Delete, Insert and Update toolbar buttons at the top of the window to add SQL statements to your ETL.
8. To run the ETL as a stored procedure (that already exists in the data warehouse):
   a. Select the Execute as Stored Procedure check box.
   b. Click the Stored Procedure toolbar button.
   c. Replace STORED_PROCEDURE with the name of your stored procedure and replace (PARAM1, PARAM2) with any parameters that it needs. Note that parameters must be separated by a comma. If no parameters are required, use empty parenthesis or drop them altogether.
9. Use the Undo, Redo and Reset buttons at the bottom of the window if needed.
10. Optionally, specify a description in the Description box at the bottom of the window.
11. To save your ETL, click OK.

Single table example
The following example, based on the Data warehouse project tutorial (page 100) in Getting started with Data Warehouse projects (page 97), demonstrates how to concatenate two columns called "First Name" and "Last Name" into a single column called "FullName".

1. Click the Manage button in the Model panel. The Manage Model window opens.
2. Select Employees from the Entities list on the left.
3. Click the + (plus) toolbar button to add a new Attribute. A new row is added to the Attributes table.
4. Type any letter in the Column Name column to bring up the "Add New" option. Click the "Add New" option when it appears.
   The New Attribute Domain window opens.
5. In the Name field, type FullName. From the Type drop-down list, select Varchar. In the Length field, enter 100.
6. In the History column, select type 1 from the drop-down list.
7. Click OK to close the New Attribute Domain window and add the attribute to the Attributes table.
8. Then click OK again (below the newly added attribute) to exit edit mode.
10. In the Data Warehouse panel, click the Create button.
11. After the Data Warehouse tables have been created, close the Creating Data Warehouse window.
12. Click the Manage button in the bottom left corner of the Data Warehouse panel. The Manage Data Warehouse Tasks window opens.
13. To view the current mappings between the source columns and data warehouse columns, click the Map_Employees_1 link in the Mappings column. A "Processing" icon is displayed while the mappings are generated. After the mappings are generated, the Edit Mappings - Map_Employees_1 window opens automatically. Note that the FullName column has been added to the data warehouse columns, but is currently not mapped to the source columns.

14. The next stage is to define an ETL that will map the First Name and Last Name source columns to the FullName data warehouse column.
15. Close the Edit Mappings - Map_Employees_1 window and then select the Single Table ETL tab on the left.
16. Select Employees in the Entity column and then click the New button above the column. The Add New Single Table ETL window opens.
17. Specify a name or leave the default name and then click OK.
18. Click the Edit button (represented by a pencil icon) at the end of the Employees row. The Edit Single Table ETL: <Name> window opens.
19. In the editing pane on the right, enter the following instruction:
   UPDATE dbo.TSTG_EMPLOYEES set
   FullName = LASTNAME + FIRSTNAME
20. Click OK to save the ETL and close the window.
After Compose has finished populating the Data Warehouse, you can open the table in Microsoft SQL Server Management Studio and verify that the new column has been added with the correct data.

ETL execution sequence

The execution sequence of ETL scripts in Data Warehouse projects should be taken into consideration when writing and ordering the scripts. A proper understanding of the ETL execution order is important for preventing errors, such as those resulting from actions being performed on objects that do not yet exist.

ETL scripts are usually executed in the following order:

Custom Pre-Loading (Source of data: landing tables)
  ↓
Mappings (Source of data: landing table)
  ↓
Multi Table ETL (Source of data: staging tables)

Whereas it's possible to define a single table ETL in the Multi Table ETL script, the advantage of defining it as a single table ETL script is that it will be able to run in parallel with other tables.

↓

Single Table ETL (Source of data: staging tables)
  ↓
Post Loading ETL (Source of data: data warehouse tables)

Within each of the above groups, the scripts are executed according to their numeric order (from lowest to highest), which is set by the user-defined Sequence Number. The execution order of several scripts in a group with the same sequence number will be random.

Managing tasks

A task contains the mappings between the columns in the landing zone tables and the columns in the logical entities. The same mappings can be used by several tasks. You can create new tasks, duplicate tasks and edit existing tasks as required.

The following options are available:

- Adding and duplicating tasks (page 188)
- Editing column mappings (page 189)
- Creating custom ETL Scripts (page 184)
You must regenerate the task and then run a data warehouse task whenever the mappings are modified or whenever custom ETLs are added or modified. Populating the data warehouse can either be done manually as described in Controlling data warehouse tasks (page 183) or automatically as described in Scheduling tasks (page 244).

If you have already run the data mart tasks, then you also need to regenerate the data mart ETLs and run the tasks again as described in Creating and managing data marts (page 211).

Adding and duplicating tasks

As the default tasks are generated automatically, there is usually no reason to manually create or duplicate a task. An exception to this is if you import your model from ERwin without first defining global mappings. In such a case, you will need to manually add the task and create the mappings.

For more information on global mappings, see Managing global mappings (page 147).

One possible reason to duplicate a task is if your model contains different types of tables and you want to manage them in separate ETLs.

Adding a new task

To add a new task:

1. Click the Manage button at the bottom left of the Data Warehouse panel. The Manage Data Warehouse Tasks window opens.
2. Click the New toolbar button. The Add New task window opens.
3. Specify a name for the task and then click OK.
4. Select the task name in the left pane and continue from Editing column mappings (page 189).

Duplicating a task

To duplicate an existing task:

1. Click the Manage button at the bottom left of the Data Warehouse panel. The Manage Data Warehouse Tasks window opens.
2. Select the task you want to duplicate and then click the Duplicate toolbar button. The Duplicate window opens.
3. Specify a Name for the new task.
4. Select a Landing Zone.
5. Optionally change the default Schema.
6. Select a Task Type according to your Replicate task type.
7. Click OK.
8. Select the task name in the left pane and continue from Editing column mappings (page 189).
Editing column mappings

The mappings show the current mapping between the landing zone tables and the logical entities. By default, the columns names and data in the source tables and the logical entities will be identical. However, you can manually change the mappings according to your needs, either by simply mapping a source column to a different data warehouse column and/or by using an expression.

To edit column mappings:

1. Click the Manage button in the Data Warehouse panel.
2. In the Manage Data Warehouse Tasks window, select the Mappings tab. Each of the logical entities has a corresponding mapping name.

3. In the Mappings column, click the mapping that you want to edit. The Edit Mapping: Name window opens.
4. Edit the mapping as described below.

Mapping a landing zone table column to a staging area table column

The mapping procedure differs depending on whether you are in Standard View or Compact View. For information on changing the view, see Changing the view (page 190).

In Standard View:

1. Hover the mouse cursor over the source column name as shown in the image below. A gray dot appears to the right of the column name.

2. Drag the mouse cursor from the gray dot to the desired column in the logical entity.

3. When the dotted line turns green (as shown below), release your mouse button.
Note that if the dotted line turns red (instead of a green), you will not be able to map the source column with the desired data warehouse column. A red dotted line indicates that the source and data warehouse column data types are incompatible with each other.

In Compact View:

1. Switch to Compact View as described in Change the view.
2. Drag the source column to the cell located to the left of the target data warehouse column.

Auto-generating mapping
Click the Auto-Map toolbar button.

Removing all mappings
Click the Reset toolbar button.

Changing the view
To change the view, click the Change View toolbar button.

Changing to a more compact view is recommended for sources tables that have numerous columns. In compact view, the table columns are organized in rows (instead of a single list), making it easier to locate source columns and map them to the desired data warehouse columns. You can also use the search box to filter out all columns that do not match the search string.

For information on creating mappings in Compact view, see Map a column in a landing zone table to a column in a staging area table.

Selecting a different source database
Select a database from the Landing Zone Database drop-down list on the left of the window.

Selecting a different source schema
Select a schema from the Schema drop-down list on the left of the window.

Changing the entity type
Select Table, View or Query on the left of the window. If you choose the Query option, see also Define a custom query.

Defining a custom query
When the entity type is set to Query, you can set a custom select query instead of using the existing source tables/views.

To set a query:

1. Click the Set Query button. The Edit Mapping Select Query: <Mapping Name> window opens.
2. Hover the mouse cursor over a table and/or a column and then click the arrow to the right of the highlighted table/column to add it to the Query.
3. Use the Select button at the top of the window to add select statements to your query. Optionally use the Undo, Redo and Clear buttons as required.
4. Click **OK** to save your settings and close the window. The query results will be displayed on the left of the **Edit Mappings: <Name>** window.

**Selecting a different table**

Select a table from the **Table Name** drop-down list on the left of the window.

**Seeing the data of a selected table**

Select a source table and then click the **Show Source Data** button on the left of the window.

**Creating a table-level transformation (Filter)**

1. Click the **Filter** toolbar button in the **Edit Mappings: Name** window. The Expression Builder opens.
2. Continue from **Opening the expression builder (page 169)**.

   **When creating a filter for a table, the expression should return 1 for data that you want to include and 0 for data that you want to exclude.**

   **The filter will be applied after any Data Cleansing rules that are defined.**

   **Updates to records excluded by a filter (even for records previously included by the filter) are not processed while the records are filtered out. Updating of filtered out records would only resume if the record(s) once again met the filter-in condition, but any changes made while the records were filtered out would be lost.**

**Creating a column-level transformation**

1. Hover the mouse cursor over the data warehouse column for which you want to create a transformation and then click the **fx** button that appears to its right. The Expression Builder opens.
2. Continue from **Opening the expression builder (page 169)**.

**Adding, deleting and renaming mappings**

You can add, rename and delete mappings as required. For example, if you want one of the logical entities to contain columns from several tables in the landing zone, then you need to add a new mapping for each of the landing zone tables.

**When mapping "From Date" columns in the Landing Zone to the "FD" Staging column, make sure that the dates in the Landing Zone columns are not earlier than the "Lowest Date" set in the Project Settings. Otherwise, any data with a "From Date" earlier than the "Lowest Date" will be ignored.**

**If some of the dates in the Landing Zone column are earlier than the "Lowest Date" and cannot be changed in the source, either change the "Lowest Date" set in the Project Settings - or - use a transformation in Replicate to convert the source dates to dates that are within"Lowest Date" to "Highest Date" range defined in your project.**
To add, delete, and rename mappings:

1. Click the Manage button in the Data Warehouse panel. The Manage Data Warehouse Tasks window opens.
2. In the left pane, select the task you want to add, delete, or rename.
3. Select the Mappings tab.

Adding a new mapping

To add a new mapping:

1. In the Logical Entities column, select the logical entity that you want to map.
2. Click the New button above the Logical Entities column. The New Mapping window opens.
3. Optionally change the default mapping name.
4. Click OK to save the mapping.
5. Enable the mapping.

Deleting a mapping

To delete a mapping:

1. In the Mappings column, hover the mouse cursor over the mapping you want to delete.
2. Click the Delete (x) button that appears to its right.
3. Click OK when prompted to confirm the deletion.

Renaming a mapping

To rename a mapping:

1. In the Mappings column, hover the mouse cursor over the mapping you want to rename.
2. Click the Rename (A) button that appears to its right. The Rename window opens.
3. Specify a new name for the mapping and then click OK.

Handling duplicate business keys

When two or more records in the data source have the same business key, you can select the Handle Duplicates check box to prevent an error from occurring when the data warehouse task is run. When this check box is selected, Compose will only add one of the records to the data warehouse.

Since Compose randomly chooses which record to add to the data warehouse, you may want to run a data warehouse task first to see if there are any duplicate record errors. In the event that there are, you can then modify the data source to remove records that have the same business key.

You should also select the Handle Duplicates check box in the following situations:
5 Data Warehouse projects

- The Data Warehouse task type is either Full Load and Change Tables or Change Tables Only. This is because the Change Tables may contain two records with the same business key: The old record and the updated record. When the Handle Duplicates check box is selected, the updated record will always be inserted/updated to/in the data warehouse.
- When a single table in the data warehouse is derived from multiple landing zone tables, the same business key will be set for each of the mappings. To prevent an error for occurring, you need to select the Handle Duplicates check box.

Handling null updates

The default handling of null updates is set in the Advanced tab of the Settings - <ETL_Set_Name> window. For each mapping, you can override the specified default.

To do this:

1. In the Manage Data Warehouse Tasks window, select the desired mapping.
2. Click the Null Updates toolbar button.
3. Select one of the available options. For a description of the options, see Handling Null Updates.

Using lookup tables

Lookup tables are useful for replacing source data with the actual data that you want to appear in the data warehouse. For example, a lookup table could be used to replace a zip code with a full address or, conversely, to replace a full address with a zip code.

To link a lookup table column to a logical entity column:

1. Click the link to the desired task in the Data Warehouse panel. The Manage Data Warehouse Tasks window opens.
2. In the Mappings column, click the mapping for the logical entity containing the result column (with the data that you want to replace). The Edit Mapping - Name window opens.
3. Hover the mouse cursor over the relevant data warehouse column and then click the Lookup button that appears to the right of the column name. The Select Lookup Table window opens.
   a. From the Database drop-down list, select the database containing the lookup table.
      
      The database must reside in your data warehouse.
   b. From the Schema drop-down list, select the schema containing your source lookup tables.
   c. Select either Table or View according to the lookup table type.
   d. From the Table drop-down list, select the lookup table.
      The right side of the Select Lookup Table window displays the lookup table columns and their data types. To view the data in the lookup table, click the Show Lookup Data button.
   e. After you have selected the lookup table, click OK.
4. After selecting the lookup table, the Lookup Transformations - Table Name.Column Name window opens. The window is divided into the following panes:
5  Data Warehouse projects

- **Upper pane:** The upper part of the right pane (Condition) displays the condition expression, which stipulates the condition(s) for performing the lookup.

- **Lower pane:** The lower part of the right pane (Result Column) displays the column result expression, which stipulates what data to replace in the target column.

5. To change the lookup table, click the Change Lookup Table button above the lookup table columns and then perform steps a. to d. above.

6. To view the lookup table or landing table data, click the Show Lookup Data or Show Landing Data buttons respectively.

7. To specify condition(s) for performing the lookup, click the Create Expression button (which changes to Edit Expression after an expression has been created) above the Condition expression. The Condition Expression - Column Name window opens.
   You can create an expression using the landing and lookup table columns on the left.
   For an example, see Lookup example (page 195). For information on creating expressions, see Creating expressions (page 168).

8. To specify what data to replace or add if the lookup conditions are met, click the Create Expression button (which changes to Edit Expression after an expression has been created) above the Result Column expression. The Result Expression - Column Name window opens.
   You can create an expression using the landing and lookup table columns on the left.
   For an example, see Lookup example (page 195). For information on creating expressions, see Creating expressions (page 168).

9. To preview the results, click the Preview Results button.

10. Click OK to save your settings and close the Lookup Transformations - Table Name.Column Name window.

Using lookup tables that do not have a task for CDC mapping

When the Store Changes option is enabled in the Replicate task, Replicate creates Change Tables in the landing zone. These tables contain only the changes to the original data. The Compose task CDC task reads the changes from Change Tables and applies them to the target tables. However, if the landing zone contains dedicated lookup tables (i.e. tables that are not associated with any Compose task), Compose will not be able to apply changes to these tables.

There are two ways of handling such a scenario, both of which are described below.

**Method 1**
Define another Replicate task with the Apply Changes replication option enabled.

**Method 2**
1. Discover the landing site and add all the lookup tables to the Compose model without any relation to/from other entities.
2. Either, define lookups from the data warehouse hub tables to the newly added entities.
   OR
   Create relationships from the data warehouse hub tables to the newly added entities.

> Creating relationships may not be a viable option when the lookup tables are complex.
3. Define a new data warehouse Change Tables Only task that updates the lookup tables.
4. Ensure that the new task runs before the data warehouse task.

The advantage of this method is twofold: a.) All the tables used in the mappings are managed by Compose, and b.) Only one Replicate task needs to be defined (which also means that the database transaction logs are read only once). The disadvantage is that you need to ensure that the task that updates the lookup entities always runs before any data warehouse task.

Lookup example
The following example shows how a lookup table is used to concatenate a Dutch translation of the category name (located in the lookup table) to the original category name located in the landing table.

The lookup could be defined using the following expressions:

   **Meaning:** Perform the lookup only if the Category ID in the landing table and the lookup table are the same.

2. Result column expression: ${Lookup.CategoryName} + 'is' + ${Landing.CategoryName}
   **Meaning:** Add the data in the CategoryName column in the lookup table to the data in the CategoryName column in the landing table (separated by the word "is").

Example:

Assuming the result column name is "Split Name", clicking the Preview Results button would display the following table:

<table>
<thead>
<tr>
<th>Split Name</th>
<th>Category Name (Lookup)</th>
<th>Category Name (Landing)</th>
<th>Category ID (Lookup)</th>
<th>Category ID (Landing)</th>
</tr>
</thead>
<tbody>
<tr>
<td>dranken is Beverages</td>
<td>dranken</td>
<td>Beverages</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Specerijen is Condiments</td>
<td>Specerijen</td>
<td>Condiments</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>Gebak is Confectionary</td>
<td>Gebak</td>
<td>Confectionary</td>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td>Zuivelproducten is Dairy Products</td>
<td>Zuivelproducten</td>
<td>Dairy Products</td>
<td>4</td>
<td>4</td>
</tr>
<tr>
<td>Grains/Granen is Grains/Cereal</td>
<td>Grains/Granen</td>
<td>Grains/Cereal</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td>Vlees/Gevolgete is Meat/Poultry</td>
<td>Vlees/Gevolgete</td>
<td>Meat/Poultry</td>
<td>6</td>
<td>6</td>
</tr>
</tbody>
</table>

Dropping and recreating tables
You can drop and recreate tables in your data warehouse as required. If you change the model after the data warehouse tables have already been created and loaded with data, you should adjust the data warehouse to reflect the modified model (as described in Validating the data warehouse (page 208)).
5  Data Warehouse projects

Some changes however cannot be resolved by adjusting the data warehouse. In such cases, you can either revert the model to its pre-modified state or drop and (optionally) recreate the data warehouse tables.

Note that dropping and recreating tables will delete all of the data in the tables and should only be performed in lieu of a better option.

In some scenarios, you need to edit the CREATE table statements before they are run. This can be done using the Generate DDL scripts but do not run them in Project settings (page 38). For example, if your data warehouse tables contain partitions, you will need to edit the script to maintain the partitions.

To drop and recreate tables:

1. In the Data Warehouse panel, select the Drop and Recreate Tables item from the menu in the top right corner. The Drop and Recreate Tables window opens.
2. You can select to drop and/or recreate one or more of the following tables:
   - **Data Warehouse & Data Marts** - The data warehouse tables are derived from the model whereas the data mart tables are derived from the data warehouse tables.
   - **Logging** - These tables are generated when the task runs and contain logging information. By default, these tables are prefixed with the string "TLOG".
   - **Intermediate** - These tables are temporary tables that are created when the task runs. By default, these tables are prefixed with the string "TTMP".
     Intermediate tables are created dynamically and therefore cannot be recreated.
   - **Error Mart** - These are the data mart exception tables. Data that is rejected by data quality rules will be copied to tables in the specified error mart schema. See also Error Mart.
   - **Archive Tables** - These are the tables that are created when the option to archive Change Tables after the changes have been applied (to the data warehouse tables) is selected. For more information, see Defining landing zones (page 132)
3. Click OK to perform the drop and/or recreate operation.

Data profiling

Data profiling is an analysis of the candidate data sources for a data warehouse to clarify the structure, content, relationships and derivation rules of the data. In short, data profiling helps you understand your data and model it correctly.

Qlik Compose enables you to profile the data in the landing zone tables before it is loaded into the data warehouse. If you discover a problem with certain data, then you can either manually adjust the source tables or create a rule for handling the data in question.
5  Data Warehouse projects

To profile the data:

1. Click the Manage button at the bottom of the Data Warehouse panel.
2. In the Manage Data Warehouse Tasks window, click the link in the Mapping column for the table you want to profile.
3. In the Edit Mappings - <Name> window, click the Data Profiler toolbar button. The Profile <Table Name> (Landing Zone) window opens. The following columns are displayed:
   - Column Name - The name of the table column
   - Nulls - The number of null values in the column
   - Count - The number of rows in the column.
   - Count Distinct - The number of unique rows in the column.
   - Duplicates - The number of duplicate values in the column.
   Note that although Compose calculates the number of duplicate values by subtracting Count Distinct from Count, the actual number of records displayed when you click the Duplicates number will be higher. This is because Compose has no way of knowing which of the records that share the same column value are legitimate duplicates (if any). It therefore displays all records that share the same value so you can decide which of them to delete (if any). For example, in the Employees table, there may be several employees that live in London (the City column). Therefore duplicates of "London" are perfectly acceptable. However, two employees with the same phone number and a different address, for example, may indicate that the phone number in one of the records was entered incorrectly. Duplicate values are quite common and usually do not indicate a problem. Where this feature is particularly useful however, is for detecting duplicate Primary Key candidate columns.
   - Data Type - The column data type
   - Max - The highest data value
   - Max Length - The longest data value
   - Min - The lowest data value
   - Min Length - The shortest data value
4. For more information about a value, click the link in the column. A window opens showing the record(s) containing the value. To add a Data Quality rule, click the Data Quality button and continue as described in Defining and managing data quality rules (page 198).
5. To only show columns that are mapped to a logical entity column, select the Only show mapped columns check box.
6. To change the number or rows sampled, select a different value from the Rows to sample drop-down list. Note that the table may contain less rows than the selected value. The Sampled records value is the actual number of rows sampled.
7. To see all the table data, click the Show Data button.

The table's Full Load data will always be shown, even for a mapping in a Change Processing (CDC) task.
8. To recalculate the data, click the **Recalculate** button. This is useful if the data in the landing zone tables is being constantly updated (for example, due to a Replicate Change Processing task).

9. To search for a particular value, start typing the value in the **Search** box. Only values that match the search term will be shown.

### Defining and managing data quality rules

There are many definitions of data quality but data is generally considered high quality if, "they are fit for their intended uses in operations, decision making and planning." (Tom Redman<Redman, T.C. (2008). With Compose, the data must be "fit" for use in a data mart.

Compose provides two ways of ensuring data quality: Data validation and data cleansing. As opposed to data validation which usually results in data being rejected, data cleansing provides a means of replacing, modifying, or deleting incomplete, incorrect or inaccurate data.

Data that is rejected by a rule will be copied to Error Mart tables in the Error Mart schema defined in the Landing Zone database settings.

Details about rejected data can be viewed in the monitor’s **Error Mart** tab. For more information, see Viewing information in the monitor (page 240).

### Defining data cleansing rules

Qlik Compose enables you to define data cleansing rules for each of a table’s columns. Each rule consists of a data validation condition and a cleansing process that is performed as required (i.e. if the data is not valid).

*Data Cleansing rules will be applied before any filters that are defined.*

#### To add a rule:

1. Click the **Manage** button at the bottom of the Data Warehouse panel.
2. In the **Manage Data Warehouse Tasks** window, click the link in the **Mapping** column for the relevant table.
3. In the **Edit Mappings - <Name>** window, click the **Data Quality** toolbar button. The **Data Quality Rules - <Table Name>** window opens.
4. To add a new rule, click the **New** toolbar button. A row is added to the rules table.
5. In the **Name** column, specify a name for the rule.
6. From the drop-down list in the **Column** column, select the column to which the rule will be applied.
7. Hover the mouse-cursor over the **Condition** column and then click the **fx** button that appears on the right.
8. In the **Edit Condition Rule** window, create a condition (using an expression) that the data in the column must meet in order to be considered valid. For more information on creating expressions, see Opening the expression builder (page 169).
    See also Simple Example Rule (page 199) below.
9. From the drop-down list in the **If Condition is False** column, select **Cleanse Silently**.
10. Hover the mouse-cursor over the **Correction** column and then click the **fx** button that appears on the right.

11. In the **Edit Correction Rule** window, create an expression to cleanse the data. For more information on creating expressions, see *Opening the expression builder (page 169)*. See also *Simple Example Rule (page 199)* below.

12. In the **Description** column, enter a description for the rule.

13. In the **Enabled** column, select or clear the check box to enable (the default) or disable the rule respectively.

**Simple Example Rule**
The condition expression on the left stipulates that the product ID number must be less than 100. If it is greater than or equal to 100, the data will be corrected using the expression on the right.

${\text{ProductID}} < 100 \, \text{or} \, \text{${\text{ProductID}} = 100}$

**Defining data validation rules**
Qlik Compose enables you to define data validation rules that are applied to the data before it is loaded into the data warehouse. In addition to defining rules, you can also define what action should be taken when data is rejected/accepted by Compose.

**To add a rule:**

1. Click the **Manage** button at the bottom of the Data Warehouse panel.

2. In the **Manage Data Warehouse Tasks** window, click the link in the **Mapping** column for the table you want to profile.

3. In the **Edit Mappings - <Name>** window, click the **Data Quality** toolbar button. The **Data Quality Rules - <Table Name>** window opens.

   The default rule rejects primary keys that have a null value and reports the rows.

4. To add a new rule, click the **New** toolbar button. A row is added to the rules table.

5. In the **Name** column, specify a name for the rule.

6. Hover the mouse-cursor over the **Rule** column and then click the **fx** button that appears on the right.

7. In the **Edit Data Quality Rule** window create a rule using an expression. For more information on creating expressions, see *Opening the expression builder (page 169)*. See also *Simple Example Rule (page 199)* below.

8. From the drop-down list in the **Error Action** column, select one of the following actions (performed when the data does not meet the rule conditions):

   - **Reject and report** - Reject the data and send a report
   - **Reject silently** - Reject the data without sending a report
   - **Reject and abort** - Reject the data and abort the data warehouse task
   - **Accept and report** - Accept the data and send a report
5 Data Warehouse projects

- When the "report" option is selected, the row is reported to the <landing_table_name>_ex table in the data warehouse error mart.
- When there are multiple data validation rules, Compose will stop evaluating the data after the first error (and report only that error). Once the error is fixed, additional data evaluation errors may be reported for the remaining rules, each time the data is loaded.

9. In the Description column, enter a description for the rule.
10. In the Enabled column, select or clear the check box to enable (the default) or disable the rule respectively.

A rule that is defined to reject or accept a non-null value (e.g. 2) in a given column will also reject/accept NULL values that appear in the same column, but in different records. To prevent this from happening, add the following condition to the rule: "and column value is not null"

Example: `LEN(${CName})<2 and (${CName} is not null)`

Simple Example Rule
The following rule stipulates that the number of units in stock must be greater than 1.

`${UnitsInStock}>1`

Managing Data Quality rules
The following options are available for managing Data Quality rules.

Enabling/disabling a Data Quality rule
Select or clear the check box in the rule’s Enabled column.

Editing a Data Quality rule
Select the rule and edit it as described in Defining data cleansing rules (page 198) and Defining data validation rules (page 199) respectively.

Deleting a Data Quality rule
Select the rule and then click the Delete button above the rules list. When prompted to confirm the deletion, click Yes.

Searching for a Data Quality rule
Enter a search term in the Search box above the rules list.

Changing the order of Data Quality rules
The order of the rules is important since rules are applied in the order that they appear. For example, placing Reject and abort rules first will prevent other rules from being applied if the data is rejected by the Reject and abort rule.

To change the order of Data Quality rules, select the rule that you want to move and then use the arrows above the rules list to change the position of the rule.
Viewing missing references

In some cases, incoming data is dependent on or refers to other data. If the referenced data cannot be loaded for some reason, you can either decide to add the data manually or continue on the assumption that the data will arrive before it is needed.

There are two ways you can view missing references in Compose. Either via the Monitor tab in the Manage Data Warehouse Tasks window or by switching the console to Monitor view and selecting the Missing References tab. The instructions below cover both of these methods.

To check for missing references in the Manage Data Warehouse Tasks window:

1. Click the Manage button in the lower left corner of the Data Warehouse panel.
2. Select the desired task in the left side of the Manage Data Warehouse Tasks window.
3. Switch to Monitor view by clicking the Monitor tab in the top right of the Manage Data Warehouse Tasks window.
4. Click the View Missing References toolbar button. The Missing References - <task Name> window opens.
   The following information is displayed:
   - General information: The run number of the task, when it started and ended, the total number of inserts and updates, and the number of reported rows (if any).
   - Missing references information:
     - Missing Records from Entity - The name of the entity with missing reference and the number of missing references.
       To see the missing record keys for the entity, click the number in parentheses to the right of the entity name.
       The Missing Record Keys for Entity - <Entity Name> window opens showing the list of missing keys and the number of times each key is referenced per entity.
     - Referenced from Entity - The entities that are referencing the entity with missing references.
     - Via Relationship - The name of the relationship in the Model.
5. To close the window, click Close.

To check for missing references in the Compose Monitor:

1. Switch the console to Monitor View.
2. Select the desired task.
3. Click the Missing References tab below the task list.
   The following information is displayed:
   - General information: The run number of the task, when it started and ended, the total number of inserts and updates, and the number of reported rows (if any).
   - Missing references information:
     - Missing Records from Entity - The name of the entity with missing reference and the number of missing references.
To see the missing record keys for the entity, click the number in parentheses to the right of the entity name.

The Missing Record Keys for Entity - <Entity Name> window opens showing the list of missing keys and the number of times each key is referenced per entity.

- **Referenced from Entity** - The entities that are referencing the entity with missing references.
- **Via Relationship** - The name of the relationship in the Model.

4. To close the window, click Close.

Missing references example

In the following example, Orders and Disputes both reference Customers.

Orders contains seven records pointing to Mr. Brown and one record pointing to Mr. Smith. Disputes contains four records referencing Mr. Brown. Mr. Brown and Mr. Smith are "missing" from Customers.

The would be reflected as follows:

<table>
<thead>
<tr>
<th>Missing Records from Entity</th>
<th>Referenced from Entity</th>
<th>Via Relationship</th>
</tr>
</thead>
<tbody>
<tr>
<td>Customers (2)</td>
<td>Orders (8)</td>
<td>Customers</td>
</tr>
<tr>
<td>-</td>
<td>Disputes (4)</td>
<td>CustomerDisputes</td>
</tr>
</tbody>
</table>

Clicking the number to the right of Customers (in the Missing Records from Entity column) would open the following window:

<table>
<thead>
<tr>
<th>Key</th>
<th>Referenced from Entity</th>
<th>Via Relationship</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mr. Brown</td>
<td>Orders (7)</td>
<td>Customers</td>
</tr>
<tr>
<td>-</td>
<td>Disputes (4)</td>
<td>CustomerDisputes</td>
</tr>
<tr>
<td>Mr. Smith</td>
<td>Orders (1)</td>
<td>Customers</td>
</tr>
</tbody>
</table>

See also: How Compose handles missing references in the data warehouse (page 178).

Viewing and exporting task statements

You can view the task statements that were run during the data warehouse task. You can also export the task statements to a CSV file for reviewing and sharing.

To view the task statements:

1. Click the Manage button at the bottom left of the DATA WAREHOUSE panel. The Manage Tasks window open.
2. Click the Task Statements toolbar button.
3. The Task Statements - <Name> window opens in List View. Navigate through the commands using the scroll bar or find specific commands using the Search box.
OR
Click the Item View button and navigate through the commands using the navigation buttons at the bottom of the Task Statements - <Name> window.

To jump to a specific command, type the command number in the Go To field at the bottom of the window and then press [Enter].

To export the task statements to a CSV file:

1. In List View, click the Export to CSV File button located to the left of the search field.
2. A file named "<name>_ETL_Instructions.csv" will be saved to your default Downloads location or you will be prompted to save it (according to your browser settings).

Modifying task settings
For each task, you can modify the settings according to your needs.

To open the Settings window:

1. Click the Manage button in the bottom left of the DATA WAREHOUSE panel.
2. Select a task in the left panel.
3. Click the Settings toolbar button.

A window opens displaying the following tabs: General, Advanced, and Consolidation.

General Tab
In the General tab, the following settings are available:

- **Log level**: Select the log level granularity, which can be any of the following:
  - INFO (default) - Logs informational messages that highlight the progress of the ETL process at a coarse-grained level.
  - VERBOSE - Logs fine-grained informational events that are most useful to debug the ETL process.
  - TRACE - Logs finer-grained informational events than the VERBOSE level.

  The log levels VERBOSE and TRACE impact performance. Therefore, you should only select them for troubleshooting if advised by Qlik Support.

- **Default History Resolution**: Choose the granularity of the "From Date" column value when a new history record is inserted:
  - Minutes to update with the date and time. This is the default. When this option is selected, a new record will be inserted each time the data is updated.
  - Days to update with the date only. When this option is selected, only one record (the most recently updated) will be inserted at the end of the day.
These settings will be applied, regardless of the original source column (when mapped) or Change Table [header_] timestamp column (when not mapped) granularity. So, for instance, if a source column with date and time granularity is mapped to the "From Date" column and Days is selected, then only one record (the most recently updated) will be inserted at the end of the day.

- When updating a non-null data warehouse column with a null value:
  - Do not change the target value: Select this to keep values unchanged between two mappings for the same record. For instance, if the same record exists in two different source tables (A and B), but the record in Table A has a null value for data that is present in Table B (e.g. ZIP Code). In this case, if the record in Table A arrives after the record in Table B, the target value will be set to null. Selecting this option will prevent such an occurrence.

- Set the target value to null: Select this if you want the source and target values to correspond. This can be useful, for example, when a person moves address and one of the column values (e.g. "State") changes to null.

Advanced Tab

In the Advanced tab, the following settings are available:

- **Sequential Processing**: Select this option if you want all the data warehouse tasks to run sequentially, even if they can be run in parallel. This may be useful for debugging or profiling, but it may also affect performance.

- **Maximum number of database connections**: Enter the maximum number of connections allowed. The default size is 10.
  For more information, see *Determining the required number of database connections (page 22)*.

- **JVM memory settings**: Edit the memory for the java virtual machine (JVM) if you experience performance issues. Xms is the minimum memory; Xmx is the maximum memory. The JVM starts running with the Xms value and can use up to the Xmx value.

- **Position in default workflow**: Select where you want the data warehouse tasks to appear in the default workflow. For more information on workflows, see *Workflows (page 248)*.

- **Optimize for initial load**: Optimizes initial load in certain cases. Only select this option if the source tables do not reference missing records, use lookups, map different source records to the same record, do not contain Type 1 self-references, or contain historical records. Note also that when this
option is selected, the following features are not supported:

- Data quality rules
- Derived attributes
- Consolidation of uniform sources (see Consolidation below)
- The Handle duplicates option

In the event that the task is used for incremental loading (using query-based change processing), clear the check box after the initial load task completes and regenerate the task.

- **Write task statement duration to the TLOG_PROCLOG table in the data warehouse:** This option is useful for troubleshooting performance issues with ETL processes as it records the duration of each task statement in a special table (named TLOG_PROCLOG) in the data warehouse. You can then use this information to locate task statements with abnormal duration times and modify them accordingly.

- **Do not create indexes for data warehouse tables:** During the task, Compose creates an internal index for each of the Data Warehouse tables (for query optimization). When running several consecutive tasks (e.g. via a workflow) with a large volume of tables, this process can be extremely time-consuming. In such a scenario, best practice is to select the check box for each of the tasks, except the last one.

- **Do not truncate staging tables:** Select this option if you want the ETL process to preserve the staging tables. Only use for debugging.

- **Stop processing after populating the staging tables:** Select if you do not want to proceed to populating the warehouse. Only use for debugging.

- **Do not drop temporary tables:** Select this option if you want to keep the temporary tables created during the ETL process. Only use for debugging.

### Consolidation Tab

*Requires Compose August 2021 Service Release 03 or later.*

When the **Consolidate uniform sources** option is enabled, Compose will read from the selected data sources and write the data to one consolidated entity. This is especially useful if your source data is managed across several databases with the same structure, as instead of having to define multiple data warehouse tasks (one for each source), you only need to define a single task that consolidates the data from the selected data sources.

Consolidation tab showing selected data sources
Data Warehouse projects

Editing the list of data sources requires you to regenerated the task.

The list of selectable data sources reflects the list of Source Databases that appears in the Databases panel in Designer view.

To facilitate downstream processing, you might want to add a record identifier column (for example, SourceID) to the primary key of all your entities. However, if one entity references another (for example, Orders → Customers), a naming conflict will arise as the new column (SourceID) will then appear in the referencing entity (Orders) twice. To prevent such conflicts from occurring, you should add the column to each entity with a unique prefix derived from the entity name. So, continuing with our Orders → Customers relationship example, the column name in the Orders entity should be orders_SourceID while the column name in the Customers entity should be customers_SourceID.

Prerequisites

- The structure of the tables in the selected sources must be identical.
- Source type can be Table or View, but not Query.

The source data does not have to reside in tables only or in views only; it can be ingested from a combination of views and tables. For example, the source data might be ingested from tables A, B, and C in Landing 1, and views A, B, and C in Landing 2.

See also: Editing column mappings (page 189).

Limitations and considerations

- The Optimize for initial load option is not supported with consolidation.
- A selected data source cannot contain an asterisk (*) in its specified schema name (currently only Microsoft SQL Server supports asterisks in schema names).
5 Data Warehouse projects

See also: *Using Microsoft SQL Server as a source (page 136)*
- If you have existing Full Load and Change Tables (CDC) tasks, setting the consolidation settings for the Full Load task will not automatically set the consolidation settings for the Change Tables task as well. You need to do this manually.
  See also: *Adding and duplicating tasks (page 188) and Creating a Change Processing task (page 181)*
- Uniform consolidation settings will not be included in task settings that are exported to a CSV file.
  See also: *Migrating objects as CSV files (page 46)*
- Lineage and project documentation will not reflect all of the selected sources.
  See also: *Exporting project documentation (page 93) and Lineage and impact analysis (page 164).*
- Custom ETLs (Pre Loading ETL, Multi Table ETL, Single Table ETL, and Post Loading ETL) will run only once, regardless of how many sources are selected.
  See also: *Creating custom ETL Scripts (page 184).*
- Generating the ETLs will only validate the Landing Zone database(s) defined in the mappings, and not all of the data sources selected in the **Consolidation** tab.
- Error marts will be created for each Landing Zone database.

To see the number of reported rows in each error mart:

1. Open the **Manage Data Warehouse Tasks** window, and select the consolidation task in the left pane.
2. Select the **Monitor** tab, and click the **Total Reported Rows** number.

Alternatively:

1. Switch to the main Monitor view and select the consolidation task.
2. In the **Progress Status** tab (below the tasks list), click the **Total Reported Rows** number.

The **Error Mart - <task-name>** window opens.

*Error Mart window showing the number of rows reported for each error mart*

For more information on error marts, see *Viewing information in the monitor (page 240).*

Monitoring tasks with consolidates sources

The monitor shows the sum total of all the records (for example, the total number of INSERTs) from all of the selected sources.
Validating the data warehouse

Data warehouse validation should be performed each time the model is edited (after the data warehouse has already been created). Validating the data warehouse allows you automatically resolve any differences between the model and the data warehouse.

For a data warehouse to be considered valid, the tables defined in the data warehouse need to be identical to the physical tables in terms of metadata. Depending on the change, this may require adjusting the physical tables or dropping and recreating them (via Compose).

If the data warehouse is not valid, any tasks that you attempt to run will fail.

Changes to Distribution Keys cannot be validated (or adjusted). Such changes need to be applied manually to the Data Warehouse tables.

Sometimes, however, the differences between the model and the data warehouse cannot be resolved automatically. In such cases, you need to drop and recreate the tables as described in Dropping and recreating tables (page 195).
To validate the data warehouse:

1. Click the **Validate** button at the bottom right of the Data Warehouse panel. The **Validating the Data Warehouse** progress window opens.
   If any differences are detected, the following message will be displayed: *The data warehouse is different from the model.*
2. Click **Close**. The **Model and Data Warehouse Comparison Report** window opens.
3. Review the report and then click **Adjust Automatically** to resolve the differences automatically or **Generate Adjust Script** to generate a script with the adjust commands.

   **Note**: The **Adjust Automatically** button will be disabled either if the **Generate DDL scripts but do not run them** option is selected or if Compose is unable to automatically adjust the data warehouse. In such cases, you should click **Generate Adjust Script** as described below.

   - Due to Google Cloud BigQuery limitations, if Compose is unable to automatically adjust the data warehouse, then the generated script may be not valid either. Consequently, users should review the script carefully and adjust it manually (if required) before running.

   - If you clicked **Adjust Automatically**, the **Adjust Data Warehouse** progress window opens. When the "The data warehouse was adjusted successfully." message is displayed, you can close the window. Note that adjusting the data warehouse may require you to update the data mart. In such a case, an appropriate message will be displayed for each of the data marts that require updating.

**Cases where Compose is unable to automatically adjust the data warehouse are as follows:**

   - A data type change that is not supported by the database or a data type change that may result in data loss.
   - A change in an entity’s business key or distribution key.
   - An attribute’s history type is Type 2 and the satellite table number in the attribute’s settings has changed.

   - If you clicked **Generate Adjust Script**, the **Generate DDL Scripts** window opens showing the progress of the script generation.
   The generated scripts will be saved to:
   `<product_dir>\data\projects\<project_name>\ddl-scripts`

   Once the script(s) have been generated, you can close the **Generate DDL Scripts** window.
   After you close the **Generate DDL Scripts** window, the **DDL Script Files** window opens automatically displaying the generated scripts. The DDL Script Files provides a read-only view that allows you to review the scripts and download them.

   The scripts need to be executed directly in your data warehouse. Make sure that any modifications that you make to the scripts are done prior to executing them.
Clearing the data warehouse metadata cache

To improve performance when reading from the Landing Zone or from the Data Warehouse tables, Compose caches the metadata from both the Landing Zone and the Data Warehouse tables. However, synchronization issues may sometimes occur if the metadata structure of the Landing Zone or the Data Warehouse tables is altered outside of the Compose project.

If you aware of external changes to the metadata or if you notice any data synchronization anomalies, Compose enables you to clear the metadata cache, either using the UI or using the CLI.

Clearing the data warehouse metadata cache with the web console

To clear the metadata cache with the web console:

1. In the Data Warehouse panel, select **Clear Metadata Cache** from the menu in the top right corner. A progress window opens.
2. After the metadata cache has been cleared, click **Close** to exit the progress window.

For information on clearing the Landing Zone metadata cache, see Clearing the Landing Zone metadata cache (page 145).

Clearing the metadata cache with the CLI

To clear the metadata cache using the CLI:

1. From the **Start** menu, open the **Compose Command Line** console.
2. Run the following command:
   ```shell
   ComposeCli.exe connect
   ```
3. Run the following command:
   ```shell
   ComposeCli.exe clear_cache --project project_name [--type landing|storage] [--landing_zone source_name]
   ```
   *Where:*
5.8 Creating and managing data marts

This section explains how to create data marts from your data warehouse tables.

In this section:

- Adding star schemas and dimensions (page 220)
- Displaying data in a pivot table (page 217)
- Managing data marts (page 219)
- Pre and Post Loading ETLs (page 233)
- Viewing and exporting task statements (page 234)
- Validating the data mart (page 235)
- Modifying data mart settings (page 236)
- The "Obsolete" indicator (page 238)

Adding data marts and star schemas

This topic explains how to create and manage data marts and star schemas in Qlik Compose. Since a data mart is essentially a subset of the data warehouse, you can create any number of data marts according to your BI needs. You can also create multiple star schemas for a single data mart. Star schemas allow you to reuse existing dimension tables within the same data mart, thereby saving space in the data warehouse while at the same time improving query performance. For example, you could create one star schema with an Order Details fact table and Customers and Products dimensions and another star schema with the same dimensions but a different fact type (or the same fact type, but different dimensions). This also allows you to generate BI reports using different facts that share the same dimensions. Additionally, in a star schema, dimensions are linked with each other through one join path intersecting the fact table, facilitating accurate and consistent query results.

- If you edit an expression or a column lookup in a dimension, the changes will not be applied to existing data. To apply such changes, you need to reload the data (which could take some time, depending on the number of records and whether there are a lot of historical records).
- Data warehouse tasks cannot run in parallel with data mart tasks.
A new data mart should be created in the following situations:

- Setting up a Compose project for the first time.
- To serve the needs of each individual business unit (different data marts can be used to obtain specific information for various enterprise departments, such as accounting, marketing, sales, and so on).

**To create a data mart with a star schema:**

1. Click the **New** button located at the bottom of the **Data Mart** panel.
   OR
   Click the **Manage** button and then click the **New** button located at the top of the **Manage Data Marts** window. The **New Data Mart** window opens.
2. Optionally change the default name and provide a description.
3. Make sure that the **Start New Star Schema Wizard** check box is selected (the default) and then click **OK**. The **New Star Schema** wizard opens.
4. Provide a name and description (optional) for the star schema.
5. Select one of the available fact types:
   - **Transactional** - A star schema with a transactional fact table allows you to retrieve the desired data, even if a dimension table contains multiple versions of the same record. To use an example from the automotive industry, selecting "OrderDate" as the Transaction Date would allow you to generate a report for the number of customers who bought cars in New York between 2013 and 2016, even if a customer moved to a different city (which would also result in a new record being added to the Customers dimension).
   - **Aggregated** - A star schema with an aggregated fact table allows you to make aggregate calculations based on the fact table attributes. For instance, you could create an aggregated fact that shows the total freight costs per shipping region and product category. Additionally, the presence of a transaction date in the fact table makes it possible to retrieve the desired data, even if a dimension contains multiple versions of the same record. To use an example from the shipping industry, a shipper could use an aggregated fact to generate a report for the total cost of shipping rice to Australia from 2015-2016.
   - **State Oriented** - A star schema with a state oriented fact supports Type 2 columns in the fact table. This is useful in cases where the fact is not a singular event in time, but rather, consists of multiple "states" or events that occur over time. Typical example of facts with multiple states are insurance claims or flight reservations. There are also cases when the same entity is treated as both a fact and a dimension - for example, Customers. In such cases, a report could be generated that relates to the state of the fact, such as the time a claim was submitted to the time it was approved.
6. Click **Next**.
7. In the **Facts** screen, choose one fact for the star schema and then click **Next**. The **Dimensions** screen is displayed. The left pane lists the dimensions that can be selected while the right pane displays a diagram of the star schema with the selected dimensions. You can view a dimension’s lineage by selecting the desired dimension and then clicking the **Lineage** button. For more information on lineage, see **Lineage and impact analysis (page 164)**. The left pane of the **Dimensions** screen contains the following areas:
5 Data Warehouse projects

- **Existing Dimensions** - Lists the dimensions that already exist in your data mart. Note that only dimensions that are relevant to the selected fact table will be displayed.
- **Create New Dimensions** - Lists all of the dimensions that can be added to the star schema.
- **Date Dimensions** - Lists all of the Date dimensions that can be added to the star schema. Note that these dimensions will only be available for selection if you added the Date and Time entities to your model. For an explanation of how to do this, see *Adding Date and Time entities to your model (page 165).*
- **Time Dimensions** - Lists all of the Time dimensions that can be added to the star schema. Note that these dimensions will only be available for selection if you added the Date and Time entities to your model. For an explanation of how to do this, see *Adding Date and Time entities to your model (page 165).*

**When adding dimensions using the wizard, if a root dimension already exists in the data mart, any dimensions selected under the root dimension will be ignored.**

**Workaround:** Edit the dimension and delete or add columns as required.

8. Choose which dimensions to include in the star schema and then click **Next**.
9. If you chose **Star Schema with State Orientation** as your star schema type, click **Finish**. Otherwise, continue from Step *Adding data marts and star schemas (page 211)* below.
10. In the **Transaction Date** screen, choose which Transaction Date to include in the data mart fact table. Selecting a Transaction Date enables you to retrieve the required data, even if the Dimension table contains multiple versions of the same record. For example, a car salesman wants to know how many customers bought cars in New York between 2013 and 2015. Selecting *orderDate* as the Transaction Date for the *customers* Dimension would make it possible to retrieve this information, even if a customer moved to a different city (which would also result in a new record being added to the data mart).
11. If you chose **Transaction** as your star schema fact type, click **Finish**. If you chose **Aggregated** as your star schema fact type, continue from Step *Adding data marts and star schemas (page 211)* below.
12. In the **Aggregated Fact** screen:
   a. Select one or more columns from the Fact table on the left of the screen.
      
      **You can select multiple columns by holding down the [Shift] (sequential selection) or [Ctrl] (non-sequential selection) buttons while selecting the columns.**
   b. To add the column(s) to the **Group By** list on the right, either drag the columns to the list or click the arrowhead button to the left of the **Group By** list. Note that each dimension has a default "Group By" column that cannot be deleted.
   c. To add the column(s) to the **Aggregations** list on the right, either drag the columns to the list or click the arrowhead button to the left of the **Aggregations** list.
   d. To add new columns to the **Group By** or **Aggregations** list, click the **New** button above the list. In the **New column** window, specify a **Name**, **Type**, **Description** and **Aggregation** (when adding a new aggregation column) and then click **OK**. The column is added to the list.
5 Data Warehouse projects

e. To add an expression, hover the mouse cursor over the table cell in the Expression column and then click the fx button that appears on the right. The Edit Expression: <Name> window opens. For more information on creating expressions, see Creating expressions (page 168).
f. To delete a column, select the column in the list and then click the Delete button above the list.

You can select multiple columns for deletion by holding down the [Shift] (sequential selection) or [Ctrl] (non-sequential selection) buttons while selecting the columns.

See also Aggregation example (page 216).

13. Click Finish. The newly created star schema is displayed below the Star Schemas heading, as shown below.

14. Click the Create Tables toolbar button. The Creating Data Mart: Data Mart Name in Target progress window opens. Wait for the “Create Data Mart tables finished successfully.” message to be displayed and then click Close.
15. Click the **Generate** toolbar button. The **Generating Statements for Task: Data Mart Name** window opens. Wait for the "Generating Statement for Data Mart No. <number> finished successfully." message to be displayed and then click **Close**.

16. Click the **Run** toolbar button. The window switches to Monitor view and a progress bar shows the current progress in terms of percentage.

When the **Total ETL** reaches 100 percent, data mart population is complete. You can stop the task at any time by clicking the **Abort** toolbar button. This may be necessary if you need to urgently edit the task settings due to some unforeseen development. After editing the task settings, simply click the **Run** button again to restart the task.

- **Aborting a task may leave the data warehouse tables in an inconsistent state. Consistency will be restored the next time the task is run.**
- **In rare situations, the Monitor view in the Manage Data Marts window may not show any tasks initially. To remedy this, refresh the browser window.**

Other monitoring information such as the run details (i.e. the number of rows inserted/updated) and the task log files can be accessed by clicking the **Run Details** and **Log** buttons respectively. Should any errors occur, you can click the link at the end of the **Failed** bar for additional information that may help you troubleshoot the problem.

Once your data mart has been loaded with data, you can check that the required data is available for your BI tools. For more information, see *Displaying data in a pivot table (page 217).*

**Understanding star schema icons**

Compose displays various icons to indicate both the status and characteristics of the star schema tables. These icons are displayed in the table below.
5 Data Warehouse projects

<table>
<thead>
<tr>
<th>Icon</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Icon" /></td>
<td>Indicates that although the structure for the star schema has been defined, all or part of the dimension(s) and/or fact table do not physically exist in the data warehouse. Click <strong>Create Tables</strong> to create the tables and/or click <strong>Validate</strong> to see what needs to be adjusted.</td>
</tr>
<tr>
<td><img src="image" alt="Icon" /></td>
<td>Displayed when the dimension(s) and/or fact table physically exist in the data warehouse.</td>
</tr>
<tr>
<td><img src="image" alt="Icon" /></td>
<td>Indicates a conformed (shared) dimension.</td>
</tr>
<tr>
<td><img src="image" alt="Icon" /></td>
<td>Small squares indicate that there are denormalized tables under the root dimension table. Each square represents a denormalized table, so in the image on the left, the <strong>Orders</strong> root dimension has four denormalized tables. To view the denormalized table names, hover the mouse cursor over each of the squares.</td>
</tr>
<tr>
<td><img src="image" alt="Icon" /></td>
<td>Displayed when a dimension has a reference to itself.</td>
</tr>
</tbody>
</table>

**Aggregation example**

In the following example, Mike the organization’s data scientist, wants to create an aggregation table that shows the total freight costs per shipping region and product category; for example, the total cost of shipping rice to Australia in 2015.

To achieve this objective, he adds the `CategoryName` and `ShipRegion` attributes to the **Group By** list and then adds the `Freight` attribute to the **Aggregations** list. As Mike is interested in the total freight cost, he selects **SUM** as the **Aggregation Type**.
Displaying data in a pivot table

This section explains how you can use Compose to view the data in your star schema.

To view the data in a star schema:

1. Click the Manage button at the bottom of the Data Marts panel.
2. In the Manage Data Marts window, either:
   Switch to Monitor view (by clicking the monitor icon) in the top right corner.
   OR
   Remain in Design view and select a star schema.
3. Click the Pivot toolbar button. If you clicked the Pivot toolbar button in Monitor view and your data mart contains several star schemas, you will be prompted to select a star schema. The Select columns for Pivot table window opens. The drop-down list at the top of the window contains the Fact table and the Dimensions tables that were used to create the star schema. The Fact table name is prefixed with "Fct_" while dimension table names are prefixed with "Dim_".
4. Make sure that "Fct_<FactName>" is selected in the drop-down list and then select which fact column to add to the pivot table.
5. From the drop-down list, select a dimension and then select which dimension columns to add to the pivot table.

   If you added the Date and Time dimension tables to your data mart, you will be able to select.

6. Optionally, repeat Step 5 to add columns from different dimensions to the pivot table.
When the same column is included in two different dimensions, the pivot table may show incorrect data.

7. Click OK. The pivot table window opens. The names of columns that you can use to generate the data will be displayed at the top of the window.

8. To form the actual table, drag columns to the gray area below the column names (the X-axis) and to the gray area on the left of the window (the Y-axis). In the following example, the ShippedDate column has been dragged to the X-axis while the OrderID column has been dragged to the Y-axis.

In this example, the QTR column was selected from the Date dimension, allowing orders to be grouped by quarter.
9. Change the table format, set aggregation, or perform additional actions as described in the table below.

### Additional actions

<table>
<thead>
<tr>
<th>To</th>
<th>Do this</th>
</tr>
</thead>
<tbody>
<tr>
<td>Set the table format</td>
<td>From the upper drop-down list in the left of the pivot table window, choose one of the following:</td>
</tr>
<tr>
<td></td>
<td>• Table</td>
</tr>
<tr>
<td></td>
<td>• Table bar chart</td>
</tr>
<tr>
<td></td>
<td>• Heatmap</td>
</tr>
<tr>
<td></td>
<td>• Row heatmap</td>
</tr>
<tr>
<td></td>
<td>• Col heatmap</td>
</tr>
<tr>
<td></td>
<td>• Treemap</td>
</tr>
<tr>
<td>Aggregation options</td>
<td>From the lower drop-down list in the left of the pivot table window, choose one of the available options.</td>
</tr>
<tr>
<td></td>
<td>Note that additional drop-down lists may be displayed depending on the selected aggregation option. For example, when <strong>Sum over Sum</strong> is selected, two additional drop-down lists (containing column names) will appear below the aggregation options. The Sum over Sum aggregate is calculated by selecting one column from each of the drop-down lists.</td>
</tr>
<tr>
<td>Change the columns</td>
<td>Click the <strong>Customize Columns</strong> button and continue from Step 3 above.</td>
</tr>
</tbody>
</table>

10. Click **OK** to close the window.

### Managing data marts

This topic describes the following management options:
• Adding star schemas and dimensions (page 220)
• Editing star schemas (page 221)
• Editing dimensions (page 226)
• Deleting data marts, schemas and dimensions (page 233)

Adding star schemas and dimensions
A data mart can contain any number of star schemas and dimensions. You can either add dimensions when you create a new star schema or you can add them later and attach them to star schemas as needed. Regardless of how they are added, dimensions can be reused across several star schemas as necessary.

To add a star schema:

1. Either click the New Star Schema toolbar button.
   OR
   Right-click the Star Schemas or Dimensions items and select New Star Schema.
   The New Star Schema wizard opens.
2. Perform steps 4 to 13 in Adding data marts and star schemas (page 211). The star schema is added to the Star Schemas list.
3. If you already created the data mart tables (as described in Adding data marts and star schemas (page 211)), you need to create the new star schema tables in the data mart. To do this, perform the validation process described in Validating the data mart (page 235). Otherwise, perform steps 4 to 13 in Adding data marts and star schemas (page 211). If you also want to run a data mart task, perform step 16 as well.

To add a dimension:

1. Select the dimension(s) you want to add to the data mart. Then click OK. The dimension(s) are added to the Dimensions list.
2. If you already created the data mart tables (as described in Adding data marts and star schemas (page 211)), you need to create the new dimension table(s) in the data mart. To do this, perform the validation process described in Validating the data mart (page 235). Otherwise, perform steps 14 and 15 in Adding data marts and star schemas (page 211). If you also want to run a data mart task, perform step 16 as well.
3. Select the dimension(s) you want to add to the data mart. Then click OK. The dimension(s) are added to the Dimensions list.
4. If you already created the data mart tables (as described in Adding data marts and star schemas (page 211)), you need to create the new dimension table(s) in the data mart. To do this, perform the validation process described in Validating the data mart (page 235). Otherwise, perform steps 14 and 15 in Adding data marts and star schemas (page 211). If you also want to run a data mart task, perform step 16 as well.
5 Data Warehouse projects

To attach a newly added dimension to a star schema:

1. Perform Steps 1-2 described in To add a dimension: (page 220) above.
2. Select the dimension(s) you want to add to the star schema and then click the Add Dimension to Star Schema toolbar button. The Add Dimension <Name> to Start Schema window opens.
3. Select which start schema(s) you want to add the dimension to and then click OK. The dimension is attached to the selected star schema(s).
4. If you already created the data mart tables (as described in Adding data marts and star schemas (page 211)), you need to create the new dimension table(s) in the data mart. To do this, perform the validation process described in Validating the data mart (page 235). Otherwise, perform steps 14 and 15 in Adding data marts and star schemas (page 211). If you also want to run a data mart task, perform step 16 as well.

Importing dimensions

You can import dimensions from other data marts in the same project. This is especially useful in the following situations:

- Several developers are working on the same data mart, developing different complex dimensions
- When you need to use a dimension from another data mart and modify it slightly

To import dimensions

1. Click the Import Dimensions button in the lower toolbar. The Import Dimensions window opens.
2. From the Import from Data Mart drop-down list, select the data mart containing the dimensions to import.
3. Select which dimensions to import and then click OK.

Only dimensions that do not already exist in the current data mart (with same name) are available for selection.

The dimensions are imported to your data mart.

Editing star schemas

You can edit a star schema according to your needs. Editing options include adding columns, adding attributes and defining filters.

To edit a star schema (fact table):

1. Click the Manage button in the bottom left of the Data Mart panel. The Manage Data Marts window opens.
2. In the left pane, select the data mart containing the star schema you want to edit.
3. Expand the list of start schemas and select the star schema you want to edit. Then either click the Edit button in the lower toolbar or right-click the star schema and select Edit. The Edit Star Schema - Name window opens. The following tabs are displayed:
5 Data Warehouse projects

- **General** tab: In the **General** tab, you can edit the star schema name, the fact table name, the fact view name and the description.
  
  The following option is also available for transactional and aggregated facts:

  - **Update fact with changes to Type 2 data warehouse entities** - Select this option (the default) if you want the fact table to always be updated with the last record version of any Type 2 data warehouse entities the star schema contains.

**Example:**

Assuming the data warehouse has the following Type 2 entities:

- Orders
- Order Details
- Address

And the data mart consists of the following:

- Fact = Orders and Order Details
- Transaction date = Order Date in Orders
- Dimension = Address (Type 2)

Then the last version of Orders and Order Details will always used and Address will be updated according to the Order Date.

See also: *Data mart views (page 232).*

- **Logical Attributes** tab: In the **Logical Attributes** tab, you can add and delete columns, edit a column’s properties, view a column’s lineage, change the column order, and define filters. Edit the **Logical Attributes** tab according to the table below.

- **Physical Table** tab: The **Physical Table** tab provides a preview of the actual "physical" columns that will be created in the database. All editing tasks are performed in the **Logical Attributes** tab, except for defining table creation modifiers which is performed in the **Physical Table** tab.

For an explanation of how to define table creation modifiers, see *Example of a Valid Table Creation Modifier (page 225).*

- **Transaction Date** tab: The **Transaction Date** tab enables you to change the transaction date that you selected when you created the star schema.

For more information on transaction dates, see the **Transaction Date screen.**

---

*This tab will not be displayed if your Star Schema Type is "State Oriented."*
## Editing Logical Attributes

<table>
<thead>
<tr>
<th>Logical attributes editing options</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>To</strong></td>
</tr>
<tr>
<td>Add a new column</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Edit a column’s properties</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Delete a column</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>View a Column’s Lineage</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Create a filter</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td><strong>i</strong></td>
</tr>
<tr>
<td><strong>i</strong></td>
</tr>
</tbody>
</table>
5 Data Warehouse projects

<table>
<thead>
<tr>
<th>To</th>
<th>Do this</th>
</tr>
</thead>
</table>
| Create or edit an expression | Hover the mouse cursor over the desired table column and then click the fx button that appears to the right of the Expression column. The Expression Builder opens with the heading: Edit Expression - Column Name.  
For information on creating an expression, see Creating expressions (page 168). |
| Change the column order    | Select the column(s) you want to move and then click the Move Down/Move to Bottom or Move Up/Move to Top buttons as desired.                                                                                      |

Defining Fact Table Creation Modifiers
You can set table creation modifiers for individual star schema (fact) tables, thereby overriding the default settings in the project settings' Table creation modifiers tab (page 43). Table modifiers allow you to append additional table properties to the default Compose CREATE TABLE statement.

The available options are located below the Columns list in the Physical Table tab, and are as follows:

- **Project settings default** - When this option is selected (the default), the settings from the project settings' Table creation modifiers tab (page 43) will be used.
- **Custom** - This option is useful for if you need to append table creation modifiers to the default CREATE TABLE statement Compose uses for fact tables. Leveraging this option requires SQL coding knowledge.
- **Custom distribution and sort keys** - This option is useful if you only need to define custom distribution keys or sort keys for the fact table. Although this can also be done using the Custom option (see below), the Custom distribution and sort keys option is more convenient as it does not require any prior SQL coding knowledge.

  - Supported with Amazon Redshift only.
  - The default distribution key for all data warehouse tables is the ID column.

Setting table creation modifiers
By default, Compose creates tables in the data warehouse using the standard CREATE TABLE statement. However, organizations often need tables to be created with custom properties for better performance, special permissions, custom collation, and so on. For example, in Microsoft Azure Synapse Analytics, it’s possible to create a table as a HEAP, which is optimized for smaller tables. By default, Compose creates tables in Microsoft Azure Synapse Analytics as a CLUSTERED COLUMNSTORE INDEX, which offers the best overall query performance for large tables.

The procedure for settings table modifiers is as follows:

1. Open the star schema and select the Physical Table tab.
2. Select the Custom option.
3. Click the Edit button to open the Table Creation Modifier editor.
4. Enter the SQL parts you wish to append to the CREATE TABLE statement.
5. Optionally, but strongly recommended, validate the SQL in an external validation tool that supports your specific database and version.

![Compose does not provide any way of validating your SQL. Therefore, make sure to validate the SQL before deploying in a production environment.]

6. Click **OK** to close the editor and save your SQL parts.

### Example of a Valid Table Creation Modifier

In the following example, the Compose CREATE TABLE statement (rows 1-5) is appended with an SQL part instructing Compose to create the table as a HEAP (row 6).

```sql
CREATE TABLE MyTable
(
  column1 integer,
  column2 varchar(50),
)
WITH (HEAP)
```

**Setting and managing custom distribution keys for Amazon Redshift tables**

Set and manage distribution keys for Amazon Redshift Data Warehouse according to the table below.

<table>
<thead>
<tr>
<th>Distribution key procedures</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>To</strong></td>
</tr>
</tbody>
</table>
| Add a distribution key               | 1. Click the **Add Distribution Key** button. A row is added to the table displaying a drop-down list.  
                                           2. Select one of the available columns. |
| Set a distribution style             | From the **Distribution Style** drop-down, select **Even**, **Key** or **All**.  
                                           For more information on distribution styles, see:  
                                           **Distribution styles - Amazon Redshift** |
| Edit a distribution key              | 1. Double-click the row. A drop-down list will be shown in the **Column** column.  
                                           2. Select one of the available columns. |
| Delete a distribution key            | Select the distribution key and then click the **Delete** button. The key is deleted. |
| Change the position of a distribution key | Select the distribution key and then click the "Up" or "Down" buttons to move the key to the desired position. |
Setting and managing custom sort keys for Amazon Redshift tables

You can define one or more of the physical table columns as sort keys. Amazon Redshift stores your data on disk in sorted order according to the sort key. The Amazon Redshift query optimizer uses sort order when it determines optimal query plans. For guidelines on choosing sort keys, visit Choose the best sort key - Amazon Redshift.

Set and manage sort keys for Amazon Redshift Data Warehouse according to the table below.

<table>
<thead>
<tr>
<th>Sort key procedures</th>
</tr>
</thead>
<tbody>
<tr>
<td>To</td>
</tr>
<tr>
<td>Add a sort key</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>
|                      | &amp;nbsp;&amp;nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&nbsp;&n...
To edit a dimension:

1. Click the Manage button in the bottom left of the Data Mart panel. The Manage Data Marts window opens.
2. In the left pane, select the data mart containing the star schema you want to edit.
3. Expand the list of dimensions and select the dimension you want to edit. Then either click the Edit button in the lower toolbar or right-click the star schema and select Edit. The Edit Conformed Dimension - Name (or Edit Dimension - Name if the dimension has not yet been added to the data mart) window opens. The following tabs are displayed:
   - General tab: In the General tab, you can edit the dimension name, the dimension table name, the dimension view name and the description. You can also change the dimension’s history type by selecting Type 1 or Type 2 from the History Type drop-down list. For more information on changing the history type, see Understanding dimension history types (page 231). See also Data mart views (page 232).
   - Logical Attributes tab: In the Logical Attributes tab, you can add and delete columns, edit a column’s properties, view a column’s lineage, change the column order, and define filters. Edit the Logical Attributes tab according to the table below.
   - Physical Table tab: The Physical Table tab provides a preview of the actual “physical” columns that will be created in the database. All editing tasks are performed in the Logical Attributes tab, except for defining table creation modifiers which is performed in the Physical Table tab. For an explanation of how to define table creation modifiers, see Example of a Valid Table Creation Modifier (page 225).
4. Edit the Logical Attributes tab according to Editing star schemas (page 221).

You can apply or revert your changes at any time, simply by clicking the Apply or Cancel buttons respectively.

5. Click OK to close the window and save your settings or Cancel to close the window without saving your settings.
## 5 Data Warehouse projects

### Editing Logical Attributes

<table>
<thead>
<tr>
<th>To</th>
<th>Do this</th>
</tr>
</thead>
</table>
| Add a new column              | 1. Click the **New** toolbar button. The **New Column** window opens.  
|                               | 2. In the **Name** field, specify a name for the column.  
|                               | 3. From the **Type** drop-down list, select one of the available data types.  
|                               | 4. If the selected data type requires further configuration, additional fields will be displayed. For example, when **decimal** is selected, the **Length** and **Scale** fields will be displayed. Set the values as required.  
|                               | 5. Optionally specify a **Description**.  
|                               | 6. Click **OK** to add the column and close the **New Column** window.                                                                  |
| Edit a column’s properties    | 1. Double-click the row containing the column. The **Edit: Column Name** window opens.  
|                               | 2. Edit the properties as described in steps 2-6 of Add a new column above.                                                            |
| Delete a column               | Select the column(s) you want to delete (multi-selection is supported) and click the **Delete** toolbar button. The column(s) are deleted. |
| View a Column’s Lineage       | 1. Select the desired column.  
|                               | 2. Click the **Lineage** toolbar button. A windows displaying the column’s lineage is displayed. For more information about lineages, see *Lineage and impact analysis* (page 164). |
| Create a filter               | Click the **Filter** toolbar button. The Expression Builder opens with the heading: **Edit Filter - TableName**. For information on creating filters, see *Creating expressions* (page 168). |

**Using From Date (FD) and To Date (TD) columns in a filtering expression is not supported.**

**The assumption is that columns that are used in the filters do not change between different versions of the record. If this is not the case, the Full rebuild option should be selected in the Data Mart settings. This assumption is also true for relationships; for example, if a Sales record relates to Product which relates to Country, and the filter is applied to the product’s country, then the assumption is that the sale cannot change its product so that it is filtered in or out based on a new country.**
<table>
<thead>
<tr>
<th>To</th>
<th>Do this</th>
</tr>
</thead>
<tbody>
<tr>
<td>Create or edit an expression</td>
<td>Hover the mouse cursor over the desired table column and then click the <strong>fx</strong> button that appears to the right of the <strong>Expression</strong> column. The Expression Builder opens with the heading: <strong>Edit Expression - Column Name.</strong> For information on creating an expression, see <em>Creating expressions (page 168).</em></td>
</tr>
<tr>
<td>Change the column order</td>
<td>Select the column(s) you want to move and then click the <strong>Move Down/Move to Bottom</strong> or <strong>Move Up/Move to Top</strong> buttons as desired.</td>
</tr>
</tbody>
</table>

**Defining Dimension Table Creation Modifiers**

You can set table creation modifiers for individual dimension tables, thereby overriding the default settings in the project settings' *Table creation modifiers tab (page 43).* Table modifiers allow you to append additional table properties to the default Compose CREATE TABLE statement.

The available options are located below the **Columns** list in the **Physical Table** tab, and are as follows:

- **Project settings default** - When this option is selected (the default), the settings from the project settings’ *Table creation modifiers tab (page 43)* will be used.

- **Custom** - This option is useful if you need to append table creation modifiers to the default CREATE TABLE statement Compose uses for dimension tables. Leveraging this option requires SQL coding knowledge.

- **Custom distribution and sort keys** - This option is useful if you only need to define custom distribution keys or sort keys for the dimension table. Although this can also be done using the **Custom** option (see below), the **Custom distribution and sort keys** option is more convenient as it does not require any prior SQL coding knowledge.

  - **Supported with Amazon Redshift only.**
  - **The default distribution key for all data warehouse tables is the ID column.**

**Setting table creation modifiers**

By default, Compose creates tables in the data warehouse using the standard CREATE TABLE statement. However, organizations often need tables to be created with custom properties for better performance, special permissions, custom collation, and so on. For example, in Microsoft Azure Synapse Analytics, it’s possible to create a table as a HEAP, which is optimized for smaller tables. By default, Compose creates tables in Microsoft Azure Synapse Analytics as a CLUSTERED COLUMNSTORE INDEX, which offers the best overall query performance for large tables.

The procedure for settings table modifiers is as follows:

1. Open the dimension and select the **Physical Table** tab.
2. Select the **Custom** option.
3. Click the **Edit** button to open the Table Creation Modifier editor.
4. Enter the SQL parts you wish to append to the CREATE TABLE statement.
5. Optionally, but strongly recommended, validate the SQL in an external validation tool that supports your specific database and version.

Compose does not provide any way of validating your SQL. Therefore, make sure to validate the SQL before deploying in a production environment.

6. Click **OK** to close the editor and save your SQL parts.

Example of a Valid Table Creation Modifier

In the following example, the Compose CREATE TABLE statement (rows 1-5) is appended with an SQL part instructing Compose to create the table as a HEAP (row 6).

```sql
CREATE TABLE MyTable
(
  column1 integer,
  column2 varchar(50),
)
WITH (HEAP)
```

**Setting and managing custom distribution keys for Amazon Redshift tables**

Set and manage distribution keys for Amazon Redshift Data Warehouse according to the table below.

<table>
<thead>
<tr>
<th>Distribution key procedures</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>To</strong></td>
</tr>
<tr>
<td>Add a distribution key</td>
</tr>
<tr>
<td>Set a distribution style</td>
</tr>
<tr>
<td>Edit a distribution key</td>
</tr>
<tr>
<td>Delete a distribution key</td>
</tr>
<tr>
<td>Change the position of a distribution key</td>
</tr>
<tr>
<td><strong>Do This</strong></td>
</tr>
<tr>
<td>1. Click the <strong>Add Distribution Key</strong> button.</td>
</tr>
<tr>
<td>2. Select one of the available columns.</td>
</tr>
<tr>
<td>From the <strong>Distribution Style</strong> drop-down, select <strong>Even</strong>, <strong>Key</strong> or <strong>All</strong>.</td>
</tr>
<tr>
<td>For more information on distribution styles, see:</td>
</tr>
<tr>
<td><strong>Distribution styles - Amazon Redshift</strong></td>
</tr>
<tr>
<td>1. Double-click the row.</td>
</tr>
<tr>
<td>2. Select one of the available columns.</td>
</tr>
<tr>
<td>Select the distribution key and then click the <strong>Delete</strong> button. The key is deleted.</td>
</tr>
<tr>
<td>Select the distribution key and then click the &quot;Up&quot; or &quot;Down&quot; buttons to move the key to the desired position.</td>
</tr>
</tbody>
</table>
Setting and managing custom sort keys for Amazon Redshift tables
You can define one or more of the physical table columns as sort keys. Amazon Redshift stores your data on disk in sorted order according to the sort key. The Amazon Redshift query optimizer uses sort order when it determines optimal query plans. For guidelines on choosing sort keys, visit Choose the best sort key - Amazon Redshift.

Set and manage sort keys for Amazon Redshift Data Warehouse according to the table below.

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add a sort key</td>
<td>1. Select the <strong>Sort Keys</strong> tab below the Columns list.</td>
</tr>
<tr>
<td></td>
<td>2. From the <strong>Sort key style</strong> drop-down list, choose one of the following styles:</td>
</tr>
<tr>
<td></td>
<td>- <strong>None</strong> to disable the sort keys</td>
</tr>
<tr>
<td></td>
<td>- <strong>Compound</strong> to use all of the columns listed in the sort key definition, in the order they are listed</td>
</tr>
<tr>
<td></td>
<td>- <strong>Interleaved</strong> to give equal weight to each column in the sort key</td>
</tr>
<tr>
<td></td>
<td>3. Click the <strong>Add Sort Key</strong> button. A new row is added to the <strong>Sort Keys</strong> list. The <strong>Position</strong> column indicates the order of the column.</td>
</tr>
<tr>
<td></td>
<td>4. From the drop-down list in the <strong>Column</strong> column, select the desired column. The column is add to the list.</td>
</tr>
<tr>
<td></td>
<td>5. Click <strong>OK</strong> to save your settings and close the <strong>Edit Dimension/Edit Star Schema</strong> window.</td>
</tr>
<tr>
<td>Edit a sort key</td>
<td>1. Double-click the row. A drop-down list will be shown in the <strong>Column</strong> column.</td>
</tr>
<tr>
<td></td>
<td>2. Select one of the available columns.</td>
</tr>
<tr>
<td>Change the position of a sort key</td>
<td>Select the sort key you want to move and then click the up or down arrows to promote or demote the key.</td>
</tr>
<tr>
<td>Delete a sort key</td>
<td>Select the sort key you want to delete and then click the <strong>Delete</strong> button.</td>
</tr>
</tbody>
</table>

For more information about sort keys, visit: Choosing sort keys - Amazon Redshift.

Understanding dimension history types
By default, dimension tables are defined as history type 2, meaning that a new record is added each time a record is updated (as opposed to updating the same record). In the data warehouse, dimension tables always contain an object identifier column which is the table name appended with "OID". However, when a dimension table’s history type is 2, an additional "version" column is added as the dimension table’s Primary Key. This column, which is used to identify the version, has the same name as the table, but is appended with "VID".

The flowing figure shows the **Customer_VID** and **Customer_OID** (object identifier) columns in the **Customers** dimension table:
If a dimension table is defined as history type 1 and one or more of the columns in the corresponding data warehouse are defined as type 2, records in the dimension table will simply be replaced with the most up-to-date record in the corresponding data warehouse table.

**Bulk operations**

You can perform the following bulk operations on one or more data marts:

- Drop and recreate tables
- Enable/Disable the Optimize for initial load option
- Generate the data mart task

To perform bulk operations

1. Click the **Bulk Operations** toolbar button in the **Manage Data Marts** widow. The **Bulk Operations** window opens.

2. Select which operations to perform and on which data marts to perform them.
3. Click **OK**. The **Preparing All Data Marts** window opens, displaying the progress of the selected operations.
4. When the <n> data marts were prepared successfully message is displayed, click **Close**.

**Data mart views**

By default, data mart tables are created without corresponding views. Views may be useful however because they:

- Allow authorized employees to utilize the data mart for analytics while preventing changes to the actual data.
5 Data Warehouse projects

- Can be queried without needing to worry about "locking" the data.

When you create a view for a fact or dimension table, you also need to set the schema in which the view will be created.

Additionally, the schema permissions should only allow authorized employees to access the view(s).

For instructions on setting the view schema, see Modifying data mart settings (page 236).

See also Editing star schemas (page 221) and Editing dimensions (page 226).

Deleting data marts, schemas and dimensions

You can delete data marts, star schemas and dimensions that you no longer require.

Deleting a data mart
1. Select the data mart and then click the upper Delete Data Mart toolbar button.
2. Click OK when prompted to confirm the deletion.

Deleting a star schema
1. Select the star schema and then click the lower Delete toolbar button.
   OR
   Right-click the star schema and select Delete.
2. Optionally select the Drop unused dimensions check box.
3. Click OK when prompted to confirm the deletion.

Deleting a dimension
1. Select the dimension and then click the lower Delete toolbar button.
   OR
   Right-click the dimension and select Delete.
2. Click OK when prompted to confirm the deletion.

Pre and Post Loading ETLs

You can create and manage Pre and Post Loading ETLs as described in the table below.

| Common Table Expressions (CTEs) are not supported as well as some special clauses. |
ETL actions

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
</table>
| Define a Pre Loading or Post Loading ETL | 1. Select either the Pre Loading or Post Loading tab as appropriate.  
2. Click the New button above the User Defined ETL column. The Add New Pre/Post Loading window opens.  
3. Specify a name for the ETL and then click OK. The name is added as a link to the User-Defined ETL column.  
4. Click the link. The Edit ETL Instruction: Name window opens.  
5. To define an ETL:  
   a. Select a table and/or column and click the arrow to the right of the selected table/column to add it to the ETL.  
   b. Use the Select, Delete, Insert and Update quick access buttons at the top of the window to add SQL statements to your ETL.  
   c. To save your ETL, click OK.  
   d. To enable/disable the ETL, select or clear the check box to the left of it as required.                                                                                                                                                                                                |
| Rename a Pre Loading or Post Loading ETL | 1. Select either the Pre Loading or Post Loading tab as appropriate.  
2. At the end of the row containing the ETL name click the Rename button (A).  
3. Rename the ETL and then click OK to save the new name.                                                                                                                                                                                                                       |
| Edit a Pre Loading or Post Loading ETL  | 1. Select either the Pre Loading or Post Loading tab as appropriate.  
2. At the end of the row containing the ETL name click the Edit button.  
3. Edit the ETL as described in Step 5 of Define a Pre Loading or Post Loading ETL.                                                                                                                                                                                                 |

Viewing and exporting task statements

You can view the task statements that are run during the data mart task. You can also export the task statements to a CSV file for reviewing and sharing.

To view the task statements:

- Click the Task statements toolbar button. The ETL - Name window opens in List View. Navigate through the statements using the scroll bar or find specific statements using the Search box.  
  OR
- Click the Item View button and navigate through the statements using the navigation buttons at the bottom of the ETL - Name window

To jump to a specific statement, type the statement number in the Go To field at the bottom of the window and then press [Enter].
To export the task statements to a CSV file:

1. In List View, click the Export to CSV File button located to the left of the search field.
2. A file named "<name> ETL Instructions.csv" will be saved to your default Downloads location or you will be prompted to save it (according to your browser settings).

Validating the data mart

Whenever you edit a data mart, certain actions need to be performed to ensure that the data mart is valid. For a data mart to be considered valid, the tables defined in the data mart need to be identical to the physical tables in terms of metadata. Depending on the change, this may require adjusting the physical tables or dropping and recreating them (via Compose).

Additionally, the generated task statements must reflect the current state of the data mart. So, for example, if a filter or expression was added/edited, you will need to regenerate the task statements before running the data mart task.

**If the data mart is not valid, any tasks that you attempt to run will fail.**

Situations in which you need to validate the data mart and/or regenerate the task statements include:

- Each time the data warehouse is adjusted
- Each time a new dimension is added to a star schema
- Each time a new star schema is added to a data mart
- Adding or removing columns
- Changes to a dimension’s history type
- Changes to transformations (expressions/filters)
- Changes to a star schema’s transaction date
- Changes to a star schema’s aggregation type (max, min, etc.)

Note that clicking the Validate button only verifies that the table metadata is valid. In certain cases, even if the metadata is valid, Compose will prompt you to regenerate the task statements (by clicking the Generate button).

When you validate a data mart, Compose presents you with a list of operations that it needs to perform for the data mart to be valid. Examples of such operations include adding dimension and fact tables, deleting the fact table when the transaction date column has been deleted from the model, and so on. You can either click Adjust Automatically or Drop and Recreate Tables to approve the operations or click Cancel to continue working with the data mart in its present state.

**To validate (and adjust) the data mart:**

1. Click the Validate toolbar button in the Manage Data Marts window. The Validating the Data Mart progress window opens.
   If any differences are detected, the following message will be displayed:
   Data mart validation failed. The data mart is different from the model.
2. Click **Close**. The **Model and Data Mart Comparison Report** window opens.

3. Review the report and then click **Adjust Automatically** or **Drop and Recreate Tables** to resolve the differences.
   
   Either the **Adjust Data Mart** progress window opens or, if you clicked **Drop and Recreate Tables**, confirm the drop and recreate operation. When you confirm the drop and recreate operation, the **Creating Data Mart: Name** window is displayed.

4. When the "The data mart was adjusted successfully." or "The data mart has been created successfully." (in the case of drop and recreate) message is displayed, close the window.

5. Click the **Generate** toolbar button to regenerate the task statements.

---

**When a dimension’s history type has been changed directly in the data mart, the data mart validation will be successful, but you need to drop and recreate the tables by clicking the **Create Tables** toolbar button.**

---

For information on changing history types, see the **General tab** tab in the **Edit Dimensions** window.

### Modifying data mart settings

You can modify the data mart settings according to your needs.

#### To modify data mart settings:

1. In the **Manage Data Marts** window, select a data mart and click **Settings**.
   
   The **Setting - Data Mart Name** window opens. In the **General** tab, the following settings are available:

   - **Log level**: Select the log level granularity, which can be any of the following:
     
     - **INFO** (default) - Logs informational messages that highlight the progress of the ETL process at a coarse-grained level.
     - **VERBOSE** - Logs fine-grained informational events that are most useful to debug the ETL process.
     - **TRACE** - Logs finer-grained informational events than the **VERBOSE** level.
   
   The log levels **VERBOSE** and **TRACE** impact performance. Therefore, you should only select them for troubleshooting if advised by Qlik Support.

   - **Load Type**: Select **Full rebuild** to build the data mart from scratch each time or **Incremental loading** (default) to only load changes.

     **Incremental loading is not currently available for Aggregated or State Oriented fact tables.**

   - **Create tables in database** - By default, data mart tables are created in the database specified in the data warehouse connection settings. Optionally, click the browse button and select a different database.
This option is only available for Microsoft SQL Server.

- **Create tables in schema** - By default, data mart tables are created in the schema specified in the data warehouse connection settings. Optionally, specify a different schema, either by typing the schema name or by clicking the browse button and selecting one of the existing schemas. If you specify the name of a non-existing schema, Compose will create the schema automatically.

This option is only available for Microsoft SQL Server.

- **Create views in schema** - By default, data mart views are created in the schema specified in the data warehouse connection settings. Optionally, specify a different schema, either by typing the schema name or by clicking the browse button and selecting one of the existing schemas. If you specify the name of a non-existing schema, Compose will create the schema automatically, unless the database is Oracle.

If the view schema is different from the data mart schema, you need to grant the following permission:

```
Grant SELECT on DM_SCHEMA to DM_VIEW_SCHEMA WITH GRANT OPTION
```

See also Data mart views (page 232).

2. In the **Advanced** tab, the following settings are available:

- **Sequential Processing**: Select this option if you want to run all the data mart tasks sequentially, even if they can be run in parallel. This can be good for debugging or profiling.

- **Maximum number of database connections**: Enter the maximum number of connections allowed. The default size is 10.
  
  For more information, see Determining the required number of database connections (page 22).

- **JVM memory settings**: Edit the memory for the java virtual machine (JVM) if you experience performance issues. **Xms** is the minimum memory; **Xmx** is the maximum memory. The JVM starts running with the Xms value and can use up to the Xmx value.

- **Position in Default Workflow**: Set the position you want the data mart to appear in the default workflow. For more information on workflows, see Workflows (page 248).

- **Optimize for initial load**: This option is not applicable to the State Oriented and Aggregated fact types. If the "Incremental Loading" option is enabled (the default), clear the "Optimize for initial load" option after the initial load task completes and regenerate the Data Mart task. If the "Full Rebuild" option is enabled, selecting "Optimize for initial load" may accelerate the loading process.

- **Write task statement duration to the TLOG_PROCLOG table in the data warehouse**: This option is useful for troubleshooting performance issues with task statements as it records the duration of each task statement in a special table (named TLOG_PROCLOG) in the data warehouse. You can then use this information to locate task statements with abnormal duration times and modify them accordingly.
5 Data Warehouse projects

- **Do not drop temporary tables**: Select this option if you want to keep the temporary tables created during the task. Only use for debugging.
- **Enable table logging**: This option is available for Oracle only. When enabled, the data mart tables will be created with the Oracle LOGGING option enabled. Leaving this option unselected (the default) should improve performance, but in some cases DML operations will not be recorded in the redo log file. For more information on this option, refer to the Oracle online documentation.

3. Click OK.

The "Obsolete" indicator

The Obsolete indicator is used in data marts with Type 2 dimensions and State Oriented Fact tables only. These type of tables store history, so the attribute `OBSOLETE_INDICATION` will always be present in data mart tables that contain the From Date and To Date attributes.

The Obsolete indicator is used in data marts when retroactive changes are applied. When a row is added to a table for an object in which the From Date is earlier than the date of the existing row, the existing row will be earmarked as obsolete by setting the value for `OBSOLETE_INDICATION` to the current run number of the data mart task.

If no retroactive changes are used in the data marts, the value for the `OBSOLETE_INDICATION` will be 0.

Rows in a dimension that become obsolete will also enforce changes to the Fact table. The references of the Fact table to the obsolete rows is updated automatically so that the current, correct rows are referenced. This means that obsolete rows are isolated, in the sense that they can be deleted without subverting the referential integrity of the data mart.

The reason Compose does not delete these rows is for auditing purposes. For instance, consider a report that was generated in the past (using the data mart) and now contains incorrect information. Inspecting the obsolete rows may account for the discrepancy (i.e. incorrect data in the database as opposed to a deliberate attempt to manipulate the data).

5.9 Creating and managing command tasks

Command tasks enable you to incorporate custom processes into your Compose workflow. This is especially useful if you need to leverage external tools to transfer files, validate data, and so on. A Command task can run any script or executable supported by the operating system including batch files, Python scripts, PowerShell scripts, executables and so on.

In this section:

- *Defining command tasks (page 239)*
- *Managing command tasks (page 239)*
- *Controlling and monitoring command tasks (page 239)*
Defining command tasks

This section explains how to define a command task. You can define as many command tasks as you need and execute them at different stages of a Compose workflow.

Before you define a command task, make sure that the executable or script file that you want to run resides in the following directory on the Compose server machine:

PRODUCT_DIR\data\projects\YOUR_PROJECT\scripts

To define a command task:

1. From the project drop-down menu, select Manage Command Tasks.
   The Manage Command Tasks window opens.
2. Provide a name for the task.
3. Optionally, enter a description.
4. In the Script/Executable File field, specify the name of the files that you want to run.
5. In the Parameters field, specify any parameters required by the command. Parameters should be separated by a space.
6. The user context is the user account under which the Task will run. To change the current user context, provide the User, Password and Domain of the account under which you want the Task to run.
7. Click Save to save your changes or Discard to discard any unsaved changes.
   The task will be added to the list of tasks in the left of the window.

Managing command tasks

The table below describes the task management options.

Editing a task

Select the task in the tasks list in the left of the Manage Command Tasks window and edit it as described in Defining command tasks (page 239).

Deleting a task

Select the task in the tasks list in the left of the Manage Command Tasks window and then click the Delete toolbar button. When prompted to confirm the deletion, click OK.

Searching for a task

Enter part of the task name in the search box above the task list. The list of tasks will be filtered to show only tasks that include the search term in their name.

Controlling and monitoring command tasks

Command Tasks can be run from the Manage Command Tasks window or from the main Compose Monitor view. Although they can be run individually, command tasks are usually run as part of a workflow.
For information on defining workflows, controlling and monitoring tasks, and controlling and monitoring workflows, see Controlling and monitoring tasks and workflows (page 240).

To run a command task from the Manage Command Tasks window:

1. Open the Manage Command Tasks window and select the task you want to run.
2. Click the Run toolbar button.
3. The Manage Command Tasks window switches to Monitor view.
   In Monitor view the following information is available:
   - The task ID
   - The current status
   - When the task started and ended
   - The overall task progress

5.10 Controlling and monitoring tasks and workflows

The Compose monitor shows the current status of all your tasks and enables you to drill-down for additional information about each task. Task instances can be run immediately or scheduled to run in the future (either once or at set intervals).

In this section:

- Viewing information in the monitor (page 240)
- Viewing missing references (page 242)
- Controlling tasks (page 244)
- Notifications (page 246)
- Workflows (page 248)
- Monitoring and controlling Qlik Replicate tasks (page 252)

Viewing information in the monitor

As well as providing a high-level summary of all your tasks, the monitor also lets you view more detailed information about specific tasks.

To switch to monitor view:

1. Open a data warehouse project and click the Monitor icon in the top right of the console.
   A list of tasks is displayed for the current project. The left pane of the monitor allows you to filter the task list by status as well as indicating the current number of running, failed and completed tasks.
For each task, the monitor displays the following information:

- **Status** - Running, Completed or Failed
- **Name** - The task name.
- **Type** - Data Warehouse, Data Mart, Workflow, Command Task or Replicate Task.
- **Started and Ended** - The date and time the task started and completed (according to the server time). If the task is running, the **Ended** column will display the current progress. In the case of a Replicate task performing Change Processing, **Running CDC** will be displayed.
- **Next Instance** - The next time the task is due to run (if the task is scheduled).
- **Elapsed Time** - The time it took for the task to complete or - if the task is still running - how long the task has been running.
- **Inserted Rows** - The number of rows inserted into the data warehouse or data mart.
- **Updated Rows** - The number of rows updated in the data warehouse or data mart.
- **Reported Rows** - The number of rows reported to the error mart table(s). - The number of rows reported to the error mart table(s).

For more information on error marts, see *Defining and managing data quality rules* (page 198).

- **Disabled** - Whether the scheduled job has been disabled.

2. To view additional information about a task, select the task. The information is displayed in the following tabs in the lower pane of the monitor:

- **Details** - Use the navigation arrows to browse through task instances. For each entity, the total number of inserted, updated and deleted rows is shown.
- **History** - The **History** tab provides a list of previous task instances. To view a task instance's log file, select the task and click the **View Log** button. To view more details about a task instance, either double-click the instance or select the instance and then click the **View Instance Details** button. The **Details** tab is shown.
5 Data Warehouse projects

- **Progress Status** - The **Progress Status** tab shows the task's current progress as well as the sub-status of task statements within the task (Waiting/Standby, Running, Failed, etc.). To see details about a specific task statements, click the number to the right of the command status.
  For example, to view more information about an task statement with an error status, click the number to the right of Error.

- **Error Mart** - The **Error Mart** tab displays information when one or more Data Quality rules are enforced during the task (and the rules have been configured to report non-compliant data).
  The following information is available:
  
  - **Entity Name** - The name of the entity for which rule was created.
  - **Mappings** - The mapping for which the rule was created.
  - **Error Mart Table** - The name of the error mart table crated as a result of the rule being enforced.
  - **Schema** - The name of the schema containing the error mart table.
  - **Reported Rows** - The number of rows reported to the error mart. Clicking the number opens the actual error mart table as it appears in the data warehouse.

- **Missing References** - For a description of this tab, see Viewing missing references (page 242).

  See also: Defining and managing data quality rules (page 198).

3. To run a job immediately, select the task and then click the **Run** toolbar button.
4. To view a task's settings, select the task and then click the **Open** toolbar button. For more information about the settings, see Creating and managing the data warehouse (page 177) and Creating and managing data marts (page 211).

Viewing missing references

In some cases, incoming data is dependent on or refers to other data. If the referenced data cannot be loaded for some reason, you can either decide to add the data manually or continue on the assumption that the data will arrive before it is needed.

There are two ways you can view missing references in Compose. Either via the **Monitor** tab in the Manage Data Warehouse Tasks window or by switching the console to Monitor view and selecting the **Missing References** tab. The instructions below cover both of these methods.

To check for missing references in the Manage Data Warehouse Tasks window:

1. Click the **Manage** button in the lower left corner of the Data Warehouse panel.
2. Select the desired task in the left side of the Manage Data Warehouse Tasks window.
3. Switch to Monitor view by clicking the **Monitor** tab in the top right of the Manage Data Warehouse Tasks window.
4. Click the **View Missing References** toolbar button. The Missing References - <task Name> window opens.
   The following information is displayed:
5 Data Warehouse projects

- **General information:** The run number of the task, when it started and ended, the total number of inserts and updates, and the number of reported rows (if any).

- **Missing references information:**
  - **Missing Records from Entity** - The name of the entity with missing reference and the number of missing references.
    To see the missing record keys for the entity, click the number in parentheses to the right of the entity name.
    The Missing Record Keys for Entity - <Entity Name> window opens showing the list of missing keys and the number of times each key is referenced per entity.
  - **Referenced from Entity** - The entities that are referencing the entity with missing references.
  - **Via Relationship** - The name of the relationship in the Model.

5. To close the window, click Close.

To check for missing references in the Compose Monitor:

1. Switch the console to Monitor View.
2. Select the desired task.
3. Click the **Missing References** tab below the task list.
   The following information is displayed:
   - **General information:** The run number of the task, when it started and ended, the total number of inserts and updates, and the number of reported rows (if any).
   - **Missing references information:**
     - **Missing Records from Entity** - The name of the entity with missing reference and the number of missing references.
       To see the missing record keys for the entity, click the number in parentheses to the right of the entity name.
       The Missing Record Keys for Entity - <Entity Name> window opens showing the list of missing keys and the number of times each key is referenced per entity.
     - **Referenced from Entity** - The entities that are referencing the entity with missing references.
     - **Via Relationship** - The name of the relationship in the Model.

4. To close the window, click Close.

**Missing references example**

In the following example, **Orders** and **Disputes** both reference **Customers**.

**Orders** contains seven records pointing to Mr. Brown and one record pointing to Mr. Smith. **Disputes** contains four records referencing Mr. Brown. Mr. Brown and Mr. Smith are "missing" from **Customers**.

The would be reflected as follows:
5 Data Warehouse projects

Example table content

<table>
<thead>
<tr>
<th>Missing Records from Entity</th>
<th>Referenced from Entity</th>
<th>Via Relationship</th>
</tr>
</thead>
<tbody>
<tr>
<td>Customers (2)</td>
<td>Orders (8)</td>
<td>Customers</td>
</tr>
<tr>
<td></td>
<td>Disputes (4)</td>
<td>CustomerDisputes</td>
</tr>
</tbody>
</table>

Clicking the number to the right of customers (in the Missing Records from Entity column) would open the following window:

Example table content

<table>
<thead>
<tr>
<th>Key</th>
<th>Referenced from Entity</th>
<th>Via Relationship</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mr. Brown</td>
<td>Orders (7)</td>
<td>Customers</td>
</tr>
<tr>
<td>-</td>
<td>Disputes (4)</td>
<td>CustomerDisputes</td>
</tr>
<tr>
<td>Mr. Smith</td>
<td>Orders (1)</td>
<td>Customers</td>
</tr>
</tbody>
</table>

See also: How Compose handles missing references in the data warehouse (page 178).

Controlling tasks
You can run and stop tasks/workflow manually or you can schedule them to run periodically.

In this section:

- Running and aborting tasks manually (page 244)
- Scheduling tasks (page 244)
- Running tasks using the CLI (page 245)

Running and aborting tasks manually
You can run tasks manually and abort them if required.

To run a task manually:

- Select the task and click the Run toolbar button.

To abort a task:

- Select the task and click the Abort toolbar button.
  The task process is aborted. Note that aborting a task may leave the data warehouse or data mart tables in an inconsistent state. Consistency will be restored the next time the task is run.

Scheduling tasks
Scheduling tasks is a convenient way of continually updating the data warehouse and associated data mart(s). For instance, you could schedule a data warehouse task to run at 4:00 pm and then schedule a data mart task to run at 5:00 pm.
Note that as Compose currently does not provide a task-chaining option (i.e. run another task as soon as the current task completes), it may be better to schedule tasks using an external tool that supports this capability.

You can also use the command line interface (CLI) to run a task. For details, see Running tasks using the CLI (page 245).

To schedule a task:

1. Click the Schedule toolbar button.
2. In the <Name> Scheduler window, choose one of the following options from the Run Job drop-down list.
   - Once - to run the job once on a specific date and time.
   - Every - to run the job at set intervals.
   - Daily - to run the job every day at a specific time.
   - Weekly - to run the job on selected days at a specific time
   - Monthly - to run the job on the nth of every month at a specific time
   - Advanced - to use a Cron expression. For a description of allowed cron formats together with usage examples, see Cron format and examples (page 367).
3. Set the scheduling parameters according to the selected scheduling option.
4. Click OK to save your settings.
   The date and time the next instance is scheduled to run will appear in the Next Instance column.
5. To disable a scheduled job, select the task and click the Edit Scheduling toolbar button. Then, select the Disable check box in the <Name> Scheduler window.
6. To cancel a scheduled job for a task, select the task and click the Edit Scheduling toolbar button. Then, in the <Name> Scheduler window, click Delete.

Running tasks using the CLI

Before you can run a task, you must first run the connect command as described in Connecting to Qlik Compose server (page 76).

As Compose CLI requires Administrator permission, make sure to select "Run as administrator" when opening the command prompt.

The run_task command populates the data warehouse or data mart with data. The "ETL" operation can also be performed using the Run toolbar button located in Monitor view as well as in the Manage Data Warehouse Tasks and Manage Data Marts windows.

When this command succeeds, it returns 0.

Syntax:
where:
5  Data Warehouse projects

- **project** is the name of your Compose project
- **type** is the type of tables you want to populate. Specify `dw` to populate a data warehouse, `dm` to populate a data mart, or `wf` to run a workflow.
- **task** is the name of the task you want to run (when `--type dw`), the name of the data mart you want to populate (when `--type dm`), or the name of the workflow to run (when `--type wf`).
- **wait** is the wait time specified in seconds.

The command line can run in sync or async mode. A value of 0 (seconds) indicates sync mode. This means that as soon as the task finishes, the command line returns to prompt. The default mode is async, with a value of -1. This is also applied if you leave this parameter empty. Other negative values are not permitted.

Note that if **wait** is excluded from the command, the task may appear to complete successfully even if it encountered an error.

**Example:**

```
ComposeCli.exe run_task --project nw --task DWH_1 --type DW
Compose Control Program started...
Instance Number: 9
Compose Control Program completed successfully.
```

**Notifications**

You can select events, on the occurrence of which, a notification will be sent to the specified recipients.

**Adding a notification rule**

**To set a notification rule:**

1. Switch to **Monitor** view.
2. Click the downward arrow at the top left of the console and select **Notification Rules** from the drop-down menu.
   The **Notification Rules** window opens.
3. Click the **New** toolbar button.
   The **New Notification** wizard opens.
4. In the **Events** screen:
   - Specify a name for the notification
   - Choose for which type of events you want the notification to be sent, both at the task level and at the workflow level.
5. Click **Next**. In the **Recipients** screen:
   - Select **Windows Event** to send the notification to Windows Event Log and/or **Recipients** to send the notification to a list of email recipients.
   - If you selected **Recipients**, enter the recipient email address(es) in the **To**, **Cc** (optional) and **Bcc** (optional) boxes. Multiple addresses must be separated by a semi-colon.
6. Click Next. In the Message screen, optionally, edit the default notification message. You can add variables to the message by selecting the variable on the right and then clicking the arrow to the left of the variables list. The following variables are available:

### Notification message variables

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>${PROJECT}</td>
<td>The name of the Compose project in which the event occurred.</td>
</tr>
<tr>
<td>${TASK_NAME}</td>
<td>The name of the task in which the event occurred.</td>
</tr>
<tr>
<td>${INSERTED}</td>
<td>The number of rows inserted in the Data Warehouse/Data Mart.</td>
</tr>
<tr>
<td>${UPDATED}</td>
<td>The number of rows updated in the Data Warehouse/Data Mart.</td>
</tr>
<tr>
<td>${DELETED}</td>
<td>The number of rows deleted from the Data Warehouse/Data Mart.</td>
</tr>
<tr>
<td>${ERROR_CODE}</td>
<td>The error code if an error was encountered during the task.</td>
</tr>
<tr>
<td>${ERROR_DETAILS}</td>
<td>The error message if an error was encountered during the task.</td>
</tr>
<tr>
<td>${EVENT_TYPE}</td>
<td>The event type (Started, Error or Completed).</td>
</tr>
<tr>
<td>${EVENT_TYPE_DESCRIPTION}</td>
<td>-</td>
</tr>
<tr>
<td>${EVENT_TIME}</td>
<td>The date and time the event occurred.</td>
</tr>
<tr>
<td>${LINK}</td>
<td>A link to the relevant Compose project.</td>
</tr>
</tbody>
</table>

7. Click Next. In the Associate with screen, select whether to apply the rule to all tasks of to selected tasks. If you chose Selected Tasks, select which tasks to apply the rule to.

8. Click Next to see a summary of the notification settings or Finish to save your settings and exit the wizard.

9. If you clicked Next, review your settings and then click Finish to save the notification rule and exit the wizard or Prev to edit your settings. You can also click the headings on the right of the wizard to go directly to a specific window.

The notification will be added to the list of notifications in the Notification Rules window.

### Managing notification rules

In the Notification Rules window, you can edit, delete and enable/disable notification rules as described in the table below.
5 Data Warehouse projects

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delete a</td>
<td>Select the rule and then click the <strong>Delete</strong> toolbar button. When prompted to confirm the deletion, click <strong>Yes</strong>.</td>
</tr>
<tr>
<td>Rule</td>
<td></td>
</tr>
<tr>
<td>Edit a</td>
<td>Either double-click the rule you want to edit or select the rule and click the <strong>Edit</strong> toolbar button. Continue from <em>Notifications (page 246)</em>.</td>
</tr>
<tr>
<td>Rule</td>
<td></td>
</tr>
<tr>
<td>Disable a</td>
<td>Select the rule you want to disable and then either click the <strong>Disable</strong> toolbar button or clear the check box in the <strong>Enabled</strong> column.</td>
</tr>
<tr>
<td>Rule</td>
<td></td>
</tr>
<tr>
<td>Enable a</td>
<td>Select the rule you want to enable and then click the <strong>Enable</strong> toolbar button or select the check box in the <strong>Enabled</strong> column.</td>
</tr>
<tr>
<td>Rule</td>
<td></td>
</tr>
</tbody>
</table>

**Event IDs in Windows Event Log**

The table below lists the Event IDs for Compose events in Windows Event Log.

<table>
<thead>
<tr>
<th>Event ID</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>261</td>
<td>The task ended with error</td>
</tr>
<tr>
<td>400</td>
<td>The task has started</td>
</tr>
<tr>
<td>406</td>
<td>The task completed successfully.</td>
</tr>
</tbody>
</table>

*If a notification is set for several events, the event ID will be 0 for each of the events.*

*Event IDs for task states within Workflows are not supported.*

<table>
<thead>
<tr>
<th>Windows Event Log IDs</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Event ID</strong></td>
</tr>
<tr>
<td>261</td>
</tr>
<tr>
<td>400</td>
</tr>
<tr>
<td>406</td>
</tr>
</tbody>
</table>

**Workflows**

Workflows enable you to run tasks both sequentially and in parallel. You can either schedule workflows as described in *Scheduling tasks (page 244)* or run them manually using the **Run** toolbar button.

You can create your own workflow and/or use the built-in workflow. The built-in workflow enables you to run all of your data warehouse and data mart tasks as a single, end-to-end process. In a project with a single data mart, all tasks run sequentially in the default workflow, starting with the data warehouse tasks and ending with the data mart task. However, in projects with multiple data marts, the data mart tasks run in parallel, following the completion of the data warehouse tasks.

When you create your own workflow, you decide which tasks to include in the workflow and the order in which they will be run.

In this topic:
Adding and designing workflows
This section provides instructions for adding and creating workflows.

Adding a workflow

To add a workflow:

1. Switch to Monitor view by clicking the Monitor button in the top right of the Compose console.
2. Click the New Workflow toolbar button.
   The New Workflow window opens.
3. To create a workflow with all current tasks, select Create default workflow and then click OK. Otherwise, continue from Step 4 below. Separate workflows will be created for Full Load and Change Processing tasks. The default workflow cannot be edited and will appear as Default Workflow in the list of monitored tasks.

   To update the default workflow with newly added tasks, repeat steps 2-3 above.

4. Specify a name for your workflow.
5. To create a workflow based on an existing workflow, select the Duplicate from check box and then select an existing workflow from the drop-down list.
6. Click OK to save your settings.
   The <workflow_name> window opens.
7. Continue from Designing a workflow below.

Designing a workflow

The workflow window is divided into two panes. The pane on the left (hereafter referred to as the Elements pane) is where you design your workflow and contains two default elements: Start and End.

The Elements pane contains gateways and tasks that you can use in your workflow. The following elements are available:

- **Tasks** - All existing Data Warehouse tasks, Data Mart tasks, and Command tasks.
- **Gateways** - There are two types of gateway: Parallel Split and Synchronize. Use the Parallel Split gateway to create parallel paths. This is useful, for example, if you want two or more tasks to run in parallel. Use the Synchronize gateway to merge parallel paths. The workflow waits for all the Tasks that precede the gateway to complete before continuing the flow.
To design a workflow:

1. Drag the desired workflow elements from the Elements pane to the pane on the left.
2. Arrange the elements in the order that you want them to run.
3. Connect the elements to each other by dragging the connector from the gray dot (that appears on the right of an element when you hover the mouse cursor over it) to the target element. When a blue outline appear around the target element, release the mouse button.
4. Optionally add error paths to the workflow. The workflow will follow the error path if a task encounters an error. For example, if an error occurs with a Data Mart task, you may want to run another Data Mart task in its place.
   To add an error path, hover your mouse cursor over the task element. A red dot will appears below the element. Drag the connector from the red dot to the target element, as shown below.

Connecting two error paths to the same task should be avoided as the workflow will fail if the task tries to run twice.

Continuing a workflow in the event of parallel task failure

In a workflow, all task elements have an error port. This allows you to change the course of the workflow in the event of a task failure, as described in Step Adding and designing workflows (page 249) above. Similar to Task elements, the Synchronize gateway also has an error port which can be used to reroute the workflow if any of the tasks between the Parallel Split and Synchronize gateways should fail.

By default, a workflow will end with an error if one or more parallel tasks do not complete successfully. However, in certain cases you may want the workflow to continue, even if one or more of the parallel tasks failed.
To do this, you need to connect the error port of the relevant task(s) directly to the Synchronize gateway. You can also design the workflow so that it follows the path leading from the Synchronize error port, instead of continuing its normal flow.

In the example below, the error port of the **MyCommandTask** is connected to the Synchronize gateway, meaning that even if **MyCommandTask** task fails, the workflow will continue. However, if the **MyCommandTask** task fails, the workflow will not proceed directly to the **End** element. Instead, it will follow the Synchronize gateway’s error path to the **Source** task.

![Workflow diagram](image.png)

Validating workflows

It is strongly recommended to validate your workflow before running it. This will prevent errors from occurring during runtime due to an invalid workflow.

**Workflow rules include:**

- All elements must be connected to each other
- A workflow must contain Start and End elements and at least one task.
- A workflow cannot contain a Parallel Split gateway without a Synchronize gateway and vice versa.
- Data warehouse tasks cannot run in parallel with data mart tasks.
- Data warehouse tasks that update the same tables cannot run in parallel.
- A workflow cannot contain a Parallel Split gateway without a Synchronize gateway and vice versa.
- The execution order of elements must be sequential and not cyclic. For example an element cannot loop back to an element that precedes it in the execution order.

**To validate your workflow:**

- Click the **Validate Flow** toolbar button.

If the workflow is valid, a "**<workflow_name> is valid.**" message will be appear at the top of the window. If the workflow is not valid, a message describing the problems will appear instead.

Managing workflows

The table below describes the options available for managing workflows.
Running and monitoring workflows

You can either schedule workflows as described in *Scheduling tasks (page 244)* or run them manually using the Run toolbar button. The Run toolbar button appears both in the main Monitor view and in the workflow design window. Note that when you run a workflow from the workflow design window, a new Monitor tab is added to the window and the view automatically switches to the Monitor tab.

You can monitor the workflow either in the Monitor tab or in the Progress Status tab. During runtime, the workflow elements fill with blue providing a graphic indication of progress. If a task encounters an error, the task element will appear with red fill instead of blue.

The example below shows a workflow containing three data warehouse tasks and one data mart task. In the workflow, the data mart task will be run only after the completion of the parallel data warehouse tasks.

Monitoring and controlling Qlik Replicate tasks

Before you can create a Compose project, you need to define a Replicate task that replicates the relevant source tables from the source database to the landing zone. You can define a different task for each project or the same task can serve several projects.

Monitoring and controlling Replicate tasks from within Compose involves the following steps:
Step 1: Configure Compose to connect to the Replicate machine

1. Open the Manage Replicate Servers window using any of the following methods:
   - From the Management drop-down menu in the main toolbar, select Manage Replicate Servers.
   - In the New Data Source window, click the Replicate Server Settings link below the Associate with Replicate task field.

   The Manage Replicate Servers window opens.

2. Click Add Replicate Server.

   The Add Server window opens.

3. Enter the following information:
   - **Name**: A display name for the server.
   - **Description**: (Optional) A description for the server.
   - **Host**: The IP address or host name of the Qlik Replicate machine.

   When Replicate Server is installed on Linux, enter the IP address of the Windows machine on which the Replicate UI Server is running.

   - **Port**: Optionally, change the default port (443). You should only change the default port if you are certain that a different SSL port is being used.
   - **User Name** and **Password**: Your credentials for logging in to the Qlik Replicate machine.

   When Replicate Server is installed on Linux, enter the user name and password for the Windows machine on which the Replicate UI Server is running.

   - **Get metadata timeout**: The time to wait when discovering a task’s source database or refreshing the metadata cache before returning a timeout error.
   - **Get task timeout**: The time to wait when starting a Replicate task before returning a timeout error.

   In environments with complex networks, operations related to Replicate may exceed the default timeout limit. If you experience frequent timeouts starting tasks, discovering a task’s source database, or refreshing the metadata cache, increasing these values may help.

4. Click Test Connection and then click OK if the connection is successfully verified.

   The server is added to the Manage Replicate Servers window. Click Close to close the window.
Step 2: Add the Replicate task to the data source settings

1. Open the New Data Source or Edit Data Source: <Name> window as described in Defining landing zones (page 132).
2. Click the browse button to the right of the Associate with Replicate task field. The Select Replicate Task window opens.
3. Select a Replicate Server from the Server drop-down list. The Replicate Tasks list is populated with all tasks defined on the selected server.
4. Select the task that is replicating the source tables to the landing zone and then click OK.
5. If you want to discover source database for model generation, select Source database connection and then configure the settings as described in Defining Replicate data source connections (page 134).
6. When you’re done, click OK again to save the data source settings. The Replicate task is immediately added to the Compose monitor.

Step 3: Monitor and control the Replicate task

The example below shows how the Replicate task appears in the Compose Monitor. You can start and stop the Replicate task using the Abort and Run toolbar buttons. To view and edit the task on Replicate Server, click Open.

If a task is stopped from within Replicate, the task status in Compose will be "Completed" instead of "Aborted".

You can also define notifications for the task and add the task to a workflow. For more information, see Notifications (page 246) and Workflows (page 248) respectively.

The monitor provides various information about the task. For details, see Viewing information in the monitor (page 240).
6 Data Lake projects

This section explains how to set up Data Lake projects.

In this section:

- Defining a Qlik Replicate task (page 255)
- Adding and managing Data Lake projects (page 257)
- Getting started with Data Lake projects (page 280)
- Setting up landing and storage connections (page 283)
- Selecting source tables and managing metadata (page 294)
- Creating and Managing Storage Zone Tasks (page 315)
- Creating and managing command tasks (page 330)
- Controlling and monitoring tasks and workflows (page 331)

6.1 Defining a Qlik Replicate task

In order to work with Compose, you first need to define a Qlik Replicate task that replicates the source tables from the source endpoint to a landing zone in the storage (defined as the target endpoint in the Replicate task). The landing zone should then be defined as the data source for the Compose project.

For information on which endpoints can be used in a Replicate task that lands data for Compose, see Supported databases for Data Lake projects (page 366).

The steps below highlight the settings that are required when using Qlik Replicate with Compose. For a full description of setting up tasks in Qlik Replicate, please refer to the Qlik Replicate Help.

Prerequisites

When defining the Replicate task, make sure the following prerequisites have been met.

- If the Landing Zone database supports append, it is recommended to select Sequence as the file format in the Replicate target endpoint settings and to set the Control Tables format (if available) to Text. This will improve performance by allowing Replicate to append to the file instead of creating a new file for every Change Data Partition.
  If the above is not possible, then it is recommended to periodically delete files that are no longer required from the target directory. This will prevent files from amassing and degrading performance. This can be done automatically using Replicate's partition retention feature. For more information, see the Qlik Replicate Help.
- When Microsoft Azure HDInsight is defined as the Replicate target endpoint, you must set the endpoint's Target storage format to Sequence.
• When Oracle is defined as the source endpoint in the Replicate task, full supplemental logging should be defined for all source table columns that exist on the target and any source columns referenced in expressions.

When using live views, to ensure transactional consistency, it is recommended to turn off Speed partition mode in the Replicate task settings. When set to off, Replicate will close the partition only at the end of each transaction. This might require you to shorten the partition interval in order for the changes to be propagated to Compose in a timely manner. Shortening the partition interval might also require you to increase the partition cleanup frequency to prevent too many files from accumulating on the target and degrading performance.

For information about turning off Speed partition mode, setting partitioning intervals, and partition cleanup, see the Replicate Help.

Limitations and Considerations

• Replicate allows you to define global transformations that are applied to source/Change tables during task runtime. The following global transformations, however, should not be defined (as they are not compatible with Compose tasks):
  • Rename Change Table
  • Rename Change Table schema

• The Create target control tables in schema option in the Replicate task settings' Control Table tab is not supported.

• As Compose requires a full-after image to be able to perform Change Processing, the following Replicate source endpoints are not directly supported (as they do not provide a full-after image):
  • SAP HANA (log based)
  • Salesforce

• Compose does not support the JSON and XML data types. Therefore, columns that are usually created with these data types (by the Replicate target endpoint) should be created as STRINGs instead. This can be done automatically within Replicate using a data type transformation. For information on which target endpoints support JSON and XML data types as well as instructions on how to create a data type transformation, refer to the Replicate Help.

Setting up the task

To define the task:

1. Open Qlik Replicate and define a new task. In the New Task dialog, make sure to enable the Full Load and Store Changes options only (the Apply Changes option should not be enabled).
2. Open the Manage Endpoint Connections window and define a source and target endpoint. The target endpoint must be the Hive database where you want Compose to create the Storage Zone tables. For more information on supported endpoints, see Supported databases for Data Lake projects (page 366).
3. Add the endpoints to the Replicate task and then select which source tables to replicate.
4. To facilitate Schema evolution (page 304) in Compose, select the DDL History Control Table in the Task Settings’ Metadata|Control Tables tab. If you intend to scan all data sources (when performing schema evolution), then you must do this for ALL Replicate tasks that move data to the Landing Zone, even those with source endpoints that do not support schema evolution (e.g. Salesforce).

   If you want the DDL History Control Table to be updated with any new source tables that are added during the Replicate task, you must define Table Selection Patterns in Replicate’s Select Tables window.

5. In the Task Settings’ Store Change Setting tab, make sure that Store Changes in is set to Change tables.

6. In the Task Settings’ Change Processing|Store Changes Settings tab, enable Change Data Partitioning.

7. In the Task Settings’ Metadata|Control Tables tab, select the Change Data Partitioning Control Table.

8. Run the task.
   Wait for the Full Load replication to complete and then continue the workflow in Compose as described in Adding and managing data warehouse projects (page 35).

6.2 Adding and managing Data Lake projects
This section describes how to add and manage Data Lake projects.

Prerequisites
Before defining your Data Lake project, make sure the following prerequisites have been met.

Required clients
Depending on the Compute platform you select when you set up your project, you will need to install one of the following drivers.

- As the driver name is the same for Cloudera Data Platform, Google Dataproc, and Azure HDInsight, in order to prevent driver conflicts, only one project with any of the aforementioned compute platforms can be created per Compose installation.
- As the driver name is the same for Cloudera Data Platform and Amazon EMR, in order to prevent driver conflicts, only one project with any of the aforementioned compute platforms can be created per Compose installation.

Cloudera Data Platform, Google Dataproc, and Azure HDInsight
1. Download the Hive JDBC Driver from the Cloudera website:
   https://www.cloudera.com/downloads/
   Then, extract the HiveJDBC41.jar file from the zip file that contains the Hive JDBC Connector.
You need to register on the Simba and Cloudera websites before you can download the Hortonworks or Hive JDBC Driver.

2. Copy the HiveJDBC41.jar file to `<Compose_Installation_Dir>\java\jdbc`.
3. Restart the Qlik Compose service.

Databricks
a. Download the Databricks JDBC driver SimbaSparkJDBC42-<version>.jar from the Databricks website.
b. Copy the SparkJDBC42.jar file to `<compose_product_dir>\java\jdbc`.
c. Restart the Qlik Compose service.

Amazon EMR
a. Download the Amazon Hive JDBC Driver (HiveJDBC42.jar) from the Amazon website.
b. Copy the HiveJDBC42.jar file to `<compose_product_dir>\java\jdbc`.
c. Restart the Qlik Compose service.

Required databases and privileges
Compose Data Lake projects require four separate databases. You can create the required databases manually (which will also allow you to override the default storage location for the files) or let Compose create them for you. If you want Compose to create the databases, you need to grant the user defined in the Storage Zone settings, the CREATE DATABASE privilege for the following databases:

- **Storage Zone database** - The database specified in the Storage Zone settings. This database can have any name.
- **Landing Zone database** - The database specified in the Landing Zone settings. This database can have any name.
- **Exposed views database** - This database must have the same name as the Storage Zone database and be appended with the suffix defined in the project settings’ Naming tab (_v by default).
- **Internal views database** - Both ODS Live Views and the HDS Live Views reference this database for updates. This database must have the same name as the Storage Zone database and be appended with the suffix defined in the project settings’ Naming tab (_v_internal by default).

For more information about Compose views, see Working with views (page 291).

Required table and view privileges
The user specified in the Storage Connection Settings must be granted the following privileges on the required databases:

- SELECT
- CREATE
Data Lake project guidelines

The following topic provides guidelines for setting up your Hive cluster and determining the scheduling frequency of Change Processing storage tasks.

Working with views

Compose creates the Storage Zone with both storage tables and storage views. The storage tables are created in the database that you defined in your storage settings while two separate database are created for the views: The exposed views database and the internal views database. The exposed views database is the primary views database and contains all view types. The internal views database is used to store updates to ODS Live Views and HDS Live Views. The exposed views database and the internal views database share the same name as the Storage Zone database, but are appended with a unique suffix (by default, _v and _v_internal respectively), which is set in the project settings' Naming tab. Consumption applications should be set up to read from the exposed views database, which provide several benefits over tables including better security (requiring read-only access only), data concurrency, and minimizing duplicate records in projects defined with non-ACID storage.

Optimizing your Hive cluster setup

While the convenience of having the metadata, storage system, and compute platform on a single machine may have certain benefits (Option 1 below), it may also increase costs. Having the metadata and the storage on separate clusters to the compute platform (Option 2 below) will allow you to power down the compute machine when it’s not in use, thereby saving costs.

Compose can work with either Option 1 or Option 2 without requiring any special configuration. Simply specify the Hive server and database name in the storage connection settings.

Note that certain platforms such as Databricks automatically power the compute platform on and off as needed. With these platforms, Option 2 may not offer any benefits over Option 1.

As the “Federated” architecture may be better suited to certain environments, it’s recommended to compare the performance of both options in a test environment before deciding which one to go with.
Determining scheduling frequency

To prevent data inconsistency issues, you should schedule the Change Processing storage task frequency to be greater than or equal to the Partition every interval defined for the Replicate task (in the Task Settings’ *Store Changes Settings* tab).

As a general rule, the shorter the Change Processing task interval, the greater the impact on performance and the higher the computing costs. Therefore, it is recommended to limit the frequency of Change Processing tasks only to what is absolutely necessary.

The scheduling frequency should be determined by the rate at which data updates are required by downstream consuming applications.

Adding data lake projects

Adding a new project is the first task you need to undertake in order to work with Qlik Compose.

There are two types of project:

- Data Lake - for ingesting data from multiple sources and moving it to a storage system for analytics.
- Data Warehouse - for ingesting data from multiple sources and creating analytics-ready data marts.
This topic guides you through the steps required to set up a Data Lake project. For instructions on setting up a Data Warehouse project, see Adding data warehouse projects (page 36).

You can set up as many projects as you need, although the ability to actually run tasks is determined by your Compose license.

To prevent unpredictable behavior, each project must be defined with a dedicated Storage Zone.

To add a new Data Lake project:

1. Click the New Project toolbar button.
   The New Project wizard opens.
2. In the Project Name tab, specify the following and then click Next:
   - Name: The project name.
   - Environment Type: Optionally, change the default environment type.
   - Environment Title: Optionally, specify an environment title.
   For information about the environment settings, see Environment tab (page 266).

3. Select Data Lake as your project type.
4. Choose whether to create your Storage as an Operational Data Store (ODS) or as an Operational and Historical Data Store (ODS + HDS). Choose Operational Data Store to create an ODS from the source data or Operational and Historical Data Store to create an ODS from the source data while maintaining previous versions of updated records. Once you have made your choice, click Next.
5. In the Deployment tab, select where you want your Data Lake to be deployed. Then click Next.

   Your choice will determine which storage system options are available in the Storage screen.

6. In the Storage tab, select a storage system. If you select File System, choose a file format. Then click Next.

   • Renaming a column in Parquet or Avro format will cause the loss of all data in that column.
   • Parquet and Avro formats do not allow spaces in Primary Key column names. If your project is set up to ingest tables from Replicate, you can define a global transformation in Replicate to remove spaces from Primary Key column names.
7. In the **Compute** tab, select a compute platform and then click **Finish** to exit the wizard.

   Before configuring connectivity, make sure to install the relevant driver for your compute platform. See Prerequisites (page 257) for more information.

8. The project panels will be displayed.

9. Add a Storage Zone (Data Lake) and at least one source database as described in *Defining a Storage Zone (page 284)* and *Defining Landing Zones (page 292)* respectively.

10. Select the source tables as described in *Selecting source tables and managing metadata (page 294).*

11. Create the storage tables as described in *Creating and Managing Storage Zone Tasks (page 315).*

### Managing and monitoring projects

The table below describes the available project management options.

---

**Project management actions are performed in the main Compose window. To switch from a specific project to the main window, click the downward arrow to the right of the project name and then select All Projects from the drop-down menu.**

---

**Project management procedures**

<table>
<thead>
<tr>
<th>To</th>
<th>Do this</th>
</tr>
</thead>
<tbody>
<tr>
<td>Edit a project</td>
<td>Any of the following:</td>
</tr>
<tr>
<td></td>
<td>- Double-click the project.</td>
</tr>
<tr>
<td></td>
<td>- Right-click the project and select <strong>Designer</strong>.</td>
</tr>
<tr>
<td></td>
<td>- Select the project and then click the <strong>Open</strong> toolbar button.</td>
</tr>
</tbody>
</table>
To | Do this
---|---
Monitor a project | Any of the following:
  * Right-click the project and select **Monitor**.
  * Double-click the project and select the **Monitor** tab on the right of the console.

Create a deployment package | Any of the following:
  * Right-click the project and select **Create Deployment Package**.
  * Select the project and then select **Create Deployment Package** from the **Deployment** toolbar menu.

See also: *Project deployment (page 45)* (Data Warehouse projects) and *Project deployment (page 269)* (Data Lake projects).

Delete a project | Any of the following:
  * Right-click the project and select **Delete**.
  * Select the project and then click the **Delete** toolbar button.

View or change user permissions | Right-click the project and select **User Permissions**.

See also: *User permissions (page 348)*.

Project settings

You can modify the project settings according to your needs.

To open the project settings window

1. Open your project as described in *Managing and monitoring projects (page 262)*.
2. Click the downward arrow to the right of the project name and select **Settings** from the drop-down menu.

The **Settings** window opens.

The project settings window is divided into the following tabs:
  * General tab (page 263)
  * Naming tab (page 265)
  * Environment tab (page 266)

General tab

In this tab, the following settings are available:
Project details
Read-only information about the project deployment type, storage type, and compute type (which were all set in the project setup wizard).

Miscellaneous

- **Generate DDL scripts but do not run them:** By default, Compose executes the CREATE, ADJUST and DROP statements immediately upon user request. When you select this option, Compose will only generate the scripts but not execute them. This allows you to review and edit the scripts before they are executed.
  For example, if you want to apply custom sorting or special formatting, you will need to edit the CREATE statement accordingly.
  Note that if you select this option, you will need to copy the scripts to your Storage Zone and run them manually. You can view, copy and download the DDL scripts as described in *Viewing and downloading DDL scripts* (page 277).

  When this option is selected, you need to do the following to see the results:

  - **After running the scripts, clear the metadata cache as described in Clearing the metadata cache (page 328).**
  - **When this option is selected, you need to press [F5] (i.e. refresh the page) in order for the web console to display the updated list of tables. This can be done either before running the scripts (recommended) or after running the scripts. Note that until you refresh the browser, the information in the web console will only be partially updated.**

- **Ignore Mapping Data Type Validation:** By default, Compose issues a validation error when a landing table is mapped to a staging table with a different data type. You can select this option to allow the mapping of different data types. Note that you should only select this option if you need to map landing table data types to compatible (though not identical) staging table data types.

- **Do not display the default workflows in the monitor:** Select this option if you want to prevent the default workflows from being executed.

Default data store type for new entities
Choose one of the following:

- **Operational Data Store** - This will create an ODS from the source data.
- **Operational and Historical Data Store** - This will create an ODS from the source data while maintaining previous versions of updated records.

Deleted records in ODS (Operational Data Store) views

Requires Compose August 2021 Service Release 03 or later.

You can choose to exclude/include records marked as deleted from/in the ODS views.
- **Exclude the corresponding record from the ODS views** - This is the default option as records marked as deleted should not usually be included in ODS views.

- **Include the corresponding record in the ODS views** - Although not common, in some cases, you might want include records marked as deleted in the ODS views in order to analyze the number of deleted records and investigate the reason for their deletion. Also, regulatory compliance might require you to be able to retrieve the past record status (which requires change history as well).

**Dates**

- **Lowest Date** - The value of the "From Date" column when no other value is available.

- **Highest Date** - The value of the "To Date" column when no other value is available.

For a description of the "From Date" and "To Date" columns, see **Naming tab** below.

**Naming tab**

In this tab, you can change the default column, prefix, and suffix names.

If you change the prefix or suffix of existing tables (e.g. data warehouse tables), you need to drop and create the data warehouse and data mart tables.

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Suffix for exposed views database</td>
<td>The suffix used to identify the database used for exposed views.</td>
</tr>
<tr>
<td>Suffix for internal views database</td>
<td>The suffix used to identify the database used for internal views.</td>
</tr>
<tr>
<td>Suffix for Replicate Change Tables</td>
<td>The suffix used to identify Replicate Change Tables in the landing zone of the Storage Zone.</td>
</tr>
<tr>
<td>Prefix for storage tables</td>
<td>The prefix to add to table names in the Storage Zone. Changing this after the Storage Zone tables have already been created requires you drop and recreate your Storage Zone tables.</td>
</tr>
<tr>
<td>Prefix for all storage view types</td>
<td>The prefix to add to view names in the Storage Zone. Changing this after the Storage Zone tables have already been created requires you drop and recreate your Storage Zone tables. For more information on storage view types, see Working with views (page 291).</td>
</tr>
<tr>
<td>Name</td>
<td>Description</td>
</tr>
<tr>
<td>-----------------------------------------------------------</td>
<td>--------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Suffix for ODS standard views</td>
<td>The suffix used to identify ODS standard views. For more information on storage view types, see Working with views (page 291).</td>
</tr>
<tr>
<td>Suffix for ODS Live Views</td>
<td>The suffix used to identify ODS live views. For more information on storage view types, see Working with views (page 291).</td>
</tr>
<tr>
<td>Suffix for HDS standard views</td>
<td>The suffix used to identify HDS standard views. For more information on storage view types, see Working with views (page 291).</td>
</tr>
<tr>
<td>Suffix for HDS Live Views</td>
<td>The suffix used to identify HDS live views. For more information on storage view types, see Working with views (page 291).</td>
</tr>
<tr>
<td>&quot;From Date&quot; column name</td>
<td>The name of the &quot;From Date&quot; column. This column is added to tables that contain attributes (columns) with history. The column is used to delimit the range of dates for a given record version.</td>
</tr>
<tr>
<td></td>
<td>- This name cannot be used in other columns.</td>
</tr>
<tr>
<td></td>
<td>- NULL values are not allowed in this column. If the source &quot;From Date&quot; column contains NULLs, an expression should be created to convert them to non-null values.</td>
</tr>
<tr>
<td>&quot;To Date&quot; column name</td>
<td>The name of the &quot;To Date&quot; column. This column is added to tables that contain attributes (columns) with history. The column is used to delimit the range of dates for a given record version.</td>
</tr>
<tr>
<td></td>
<td>This name cannot be used in other columns.</td>
</tr>
</tbody>
</table>

**Environment tab**

In this tab, you can specify information about your environment that will be displayed as a banner at the top of the window when you open the project.

Provide the following information and then click **OK** to save your settings.

- **Environment type**: Select one of the following types according to your environment type: Development, Test, Acceptance, Production, Other. This information will not be displayed in the banner.
- **Environment title**: Specify a title for your environment. The title will be displayed in the banner at the top of the console.
- **Project title**: Specify a title for your project. The project title will be shown in the console banner. If both an **Environment Title** and a **Project Title** are defined, the project title will be displayed to the right of the environment title.
• The **Project title** option requires Compose August 2021 Patch Release 12 or later.
• When a project is deployed to a new environment, the environment title and environment type in the new environment will not be overridden.

The following image shows the banner with both an **Environment title** and a **Project title**:

![Banner Image]

The banner text is shown without the **Environment title** and **Project title** labels. This provides greater flexibility as it allows you add any banner text you like, regardless of the actual label name. For example, specifying **Project owner**: Mike Smith in the **Project title** field, will display that text in the banner.

Creating or Dropping Storage Tables

**Limit the number of database connections to:** The higher the number of database connections, the more storage tables Compose will be able to create or drop in parallel. While increasing the default should improve performance, it may also impact other database applications. It is therefore not recommended to increase the default unless you encounter performance issues.

As environments are project-specific, the environment information can be imported to a new project, but cannot be imported to an existing project.

Task recovery

You can set SQL state classes and error codes, on the occurrence of which, a task will be retried.

You can set the following parameters:

• **Maximum retry count:** The number of times to retry a task before exiting with failure. Increasing the number of retries will impact system resources. Therefore, only increase the default value if you expect tasks to recover after the default number of retries.

• **Interval between retry attempts (sec):** The time to wait between retry attempts. Increasing the interval will consume more system resources. Therefore, only increase the default value if it is critical that the task recover as soon as possible.

• **Retry on these SQL state classes:** The default is 08 (connection exceptions). You can add additional classes as desired. Classes should be separated with a comma.

**Example:** 08,22,2F
• **Retry also on these error codes:** The default is 1205 (which occurs when a table is locked by another process). You can add additional error codes as desired. Error codes should be separated with a comma.

**Example:** 1205,2020,233

**Limitations and considerations:**
- Schema evolution retries are not supported.
- ODBC statements comprise a small part of the task execution sequence. However, as the task retry mechanism is JDBC-based, ODBC statements will not be retried even if the specified SQL state/error code is encountered.

**Task and Workflow Information Retention**
You can set the maximum number of runs to keep task and workflow logs and messages. The default is 100. Task information includes logs, the number of inserted/updated rows per table, errors, and various other runtime messages. If you find that the number of accumulated logs and messages is degrading performance, reducing this value might help.

*This setting requires Compose August 2021 SR1 or later.*

**Resetting projects**
You can reset projects as required. This can be useful during the project development stage as it allows you to easily delete unwanted project elements.

*Be careful not to reset a project and delete data in a production environment.*

**To reset a project:**
1. Open your project as described in *Managing and monitoring projects* (page 262).
2. Click the downward arrow to the right of the project name and select **Reset Project** from the drop-down menu.
   The **Reset Project** window opens.
3. Select which elements to reset according to your project type.
   - Metadata and mapping definitions
     For information on mappings, data storage tasks and metadata, see *Selecting source tables and managing metadata* (page 294).
   - Reusable transformations
     For information on the reusable transformations, see *Reusable transformations* (page 313).
   - Storage tables and files
     For information on the Storage Zone, see *Creating and Managing Storage Zone Tasks* (page 315).
   - Command tasks
     For more information, see *Creating and managing command tasks* (page 330).*
• DDL scripts
  For more information on DDL scripts, see Project settings (page 263) and Viewing and downloading DDL scripts (page 277).

4. Click Reset Project and then click Yes when prompted to confirm your request.

Project deployment

Project deployment packages can be used to back up projects or migrate projects between different environments (e.g. testing to production). As a deployment package is intended to be deployed in a new environment, it contains the Storage Zone and data source definitions, but without any passwords. The deployment package also does not contain any data from the Storage Zone, only the metadata. The deployment package also contains the project metadata and mapping information, which should be consistent with the Landing Zone tables in the new environment.

For a complete list of objects contained in the deployment package, see Exporting a project (page 271).

Creating deployment packages

This section explains how to create a project deployment package.

To create a deployment package:

1. Choose one of the following methods:
   • In the main Compose window, right-click the desired project and select Create Deployment Package from the context menu.
   • In the main Compose window, select the desired project. Then, click the Deployment toolbar button and select Create Deployment Package from the drop-down menu.
   • In the project window, select Deployment > Create Deployment Package from the project drop-down menu.

   The Create Deployment Package - <Project_Name> window opens.

2. Provide a Version number and a Description in the designated fields and then click OK.
   A ZIP file containing a JSON file (i.e. the project settings) and a readme.txt file will be saved to your browser's default download location.
   The ZIP file name is in the following format:
   <Project_Name>_deployment_<Date>_<Time>.zip

   The readme.txt file contains the following information about the deployment package: project name, export date, exporter user name, deployment version, and description.

Deploying packages

This section explains how to deploy a project deployment package. You can only deploy packages to an existing project. Therefore, before deploying a project, create a new project with the user name and password required for connecting to the Storage Zone database and the Landing Zone database (if defined) in the new environment. In addition, the Landing Zone databases in the target project must have the same display name (defined in the Compose console) as the corresponding databases in the source project. Note that as database settings are usually environment specific, the database settings in the target project will not be overwritten by those of the source project.
When deploying, Compose does not override existing connection parameters, assuming they are environment-local. This enables you to easily migrate projects from test to production, for example, without the need to change user names, passwords or IP addresses.

If preferred, you can create an empty project and provide the required credentials after the deployment completes. In this case, an error message prompting you for the missing credentials will be displayed after the deployment completes.

To deploy a project:

1. Copy the ZIP file created when you created your deployment package to a location that is accessible from the Compose machine.
2. Open Compose and choose one of the following methods:
   - In the main Compose window, select the desired project. Then, click the Deployment toolbar button and select Deploy from the drop-down menu.
   - In the project window, select Deployment > Deploy from the project drop-down menu.
   The Deploy window opens.
3. Either drag and drop the file on the window.
   - OR-
   Click Select and browse to the location of the deployment package. In the Open window, either double-click the deployment package ZIP file or select the file and click OK.
   The package details will be displayed.
4. Click Deploy to deploy the package. When prompted to replace the existing project, confirm the operation.
   The project will be deployed.

When deploying a project defined with multiple Replicate Servers to any of the following:

- A project without any Landing Zone databases
- A project which is missing one or more Landing Zone databases defined in the source project

Then the Landing Zone settings from the source project will be used, but the missing databases will be created without a password and Replicate Server. These will need to be configured manually.

Exporting and importing projects using the CLI

Compose CLI requires Administrator permission. To grant Administrator permission, select "Run as administrator" when opening the command prompt.
Under normal circumstances, use the deployment options described in Project deployment (page 269) to export and import projects. For deployment automation or control by another tool, you can use the command line interface (CLI) to perform the following tasks:

- Exporting a project (page 271)
- Importing a project (page 272)
- Exporting the project configuration (page 273)
- Importing the Project Configuration (page 274)

To export or import a project or project configuration including passwords, you first need to change the default Master User Password.

For more information on changing the master user password, see Changing the master user password (page 31).

See also: Migrating projects from the test environment to the production environment (page 275) and Import/Export scenarios: When is a password required? (page 276)

Before running any command, you must run the Connecting to Qlik Compose server (page 76) command.

To get help when using the command line, you can run the Help command. For example, for help about exporting a project, issue the following command:

```
ComposeCli.exe export_project_repository --help
```

This brings up a list of help parameters.

### Exporting a project

You can use the Compose CLI to export a project.

Exported projects include the following:

- Data zone connections
- Metadata definitions (entities and attributes)
- Mappings between Landing Zone and Storage Zone table columns.
- Storage Zone ETL tasks
- Project settings

**Existing Storage Zone tables and generated task statements are not exported. Notifications and schedules are also not exported as they are considered to be environment-specific.**

**Syntax**

```
composecli.exe export_project_repository --project project_name --outfile output-file [--is_without_credentials] [--password password] [--master_user_password master_user_password]
```

where:
• **project** is the name of the project you want to export.
• **outfile** specifies the path to and name of the output file. This file is in JSON format. For example: C:\file.json.
• **is_without_credentials** specifies to export the project settings without the encrypted fields. When importing to a new project, you will need to manually enter the project passwords (in the Compose database connection settings) after the import completes. In addition to eliminating the need to specify a password when exporting or importing the project, the **is_without_credentials** parameter also allows the project to be used in every Compose installation, regardless of its master user password. It is also useful in the event that you would like to keep the existing passwords in the target environment (e.g. when exporting from a testing environment to an existing project in the production environment).
• **password** specifies the password for encrypting the credentials in the exported project. The **password** qualifier must be used together with the **master_user_password** qualifier described below. Use the **password** qualifier if you want to encrypt the credentials in the exported project, but do not want the source master password to be used in a different environment. The specified password must be at least 32 characters in length and can either be user-devised or generated using the `genpassword` utility described in Changing the master user password (page 31).
• **master_user_password** the master user password defined for the source machine. This must be used together with the **password** qualifier. Use the **master_user_password** qualifier if you want to encrypt the credentials in the exported project, but do not want the source master password to be used in a different environment. In such a case, when you import the project to an environment that has a different master password, you will only need to specify the **password** qualifier.

For instructions on changing the master user password, see Changing the master user password (page 31).

See also: See also: Migrating projects from the test environment to the production environment (page 275) and Import/Export scenarios: When is a password required? (page 276)

### Importing a project

You can use the Compose CLI to import a project. If you import to an existing project, all of the project settings, except the project configuration items will be overridden. For information on the project configuration items, see Exporting the project configuration (page 273).

Imported projects include the following:

- Data zone connections
- Model definitions (entities and attributes)
- Mappings between Landing Zone and Storage Zone table columns.
- Storage Zone ETL tasks

### Syntax

```
ComposeCli.exe import_project_repository --project project_name --infile input-file [--password password] [--is_without_credentials] [--override_configuration] [--dont_backup_existing_project]
```

where:
• **project** is the name of the project you want to import.
• **infile** specifies the full path to the input file (including the file name). This file is in JSON format.
  For example: C:\file.json
• **override_configuration** overrides the existing project configuration. When importing a project, the default is not to override the existing project configuration.
• **dont_backup_existing_project** specifies not to back up the existing project. By default, existing projects are backed up to the following location (and automatically restored if the import fails):
  <product_dir>\data\projects\<project_name>_backup_<timestamp>
• **is_without_credentials** specifies to import the project settings without the encrypted fields. In this case, you will need to manually enter the project passwords (in the Compose database connection settings).
• **password** the password specified with the **password** qualifier during export.
  For instructions on changing the master user password, see Changing the master user password (page 31).

**Existing Storage Zone tables and generated task statements are not imported. After the import completes, you must perform step 3 below. You may also need to perform step 1 or 2, depending on whether you changed the Storage Zone connection settings (step 1) or kept the existing connection settings (step 2).**

1. **If you changed the Storage Zone connection settings after importing the project, then you need to create the tables in the new Storage Zone.**
2. **If you edited the Metadata in a testing environment and then imported the project into a production environment, you need to validate and adjust the Storage Zone.**
3. **Generate the Data Storage task statements.**

For information on validating the Storage Zone and generating the task statements, see Creating and Managing Storage Zone Tasks (page 315).

See also: Migrating projects from the test environment to the production environment (page 275) and Import/Export scenarios: When is a password required? (page 276)

**Exporting the project configuration**

You can use the Compose CLI to export the configuration settings of an existing project. This includes Landing and Storage Connections, scheduling jobs, and notifications. This is helpful, for example, when you need to migrate configuration settings from a test environment to the production environment.

For information about migrating projects, see Migrating projects from the test environment to the production environment (page 275).

**Syntax**

```
ComposeCli.exe export_project_repository_config --project project_name --outfile output file
[--is_without_credentials] [--password password] [--master_user_password master_user_password]
```

where:
6 Data Lake projects

- **project** is the name of the project you want to export.
- **outfile** specifies the path to and name of the output file. This file is in JSON format. For example: C:\file.json.
- **is_without_credentials** specifies to export the project configuration without the encrypted fields. When importing to a new project, you will need to manually enter the Landing Zone(s) and Storage Zone passwords (in the Connections panel) after the import completes. In addition to eliminating the need to specify a password when exporting or importing the project, the **is_without_credentials** parameter also allows the project configuration to be used in every Compose installation, regardless of its Master User Password. It is also useful in the event that you would like to keep the existing passwords in the target environment (e.g. when exporting from a testing environment to an existing project in the production environment).
- **password** specifies the password for encrypting the credentials in the exported project configuration. The **password** qualifier must be used together with the **master_user_password qualifier** described below. Use the **password** qualifier if you want to encrypt the credentials in the exported project configuration, but do not want the source master password to be used in a different environment. The specified password must be at least 32 characters in length and can either be user-devised or generated using the **genpassword** utility described in Changing the master user password (page 31).
- **master_user_password** the master user password defined for the source machine. This must be used together with the **password** qualifier. Use the **master_user_password** qualifier if you want to encrypt the credentials in the exported project configuration, but do not want the source Master User Password to be used in a different environment. In such a case, when you import the project configuration to an environment that has a different Master User Password, you will only need to specify the **password** qualifier.

For instructions on changing the Master User Password, see Changing the master user password (page 31).

See also: Migrating projects from the test environment to the production environment (page 275) and Import/Export scenarios: When is a password required? (page 276)

Importing the Project Configuration

You can use the Compose CLI to import the configuration settings of an existing project. This includes Data Zone definitions, scheduling jobs, and notifications. This is helpful, for example, when you need to migrate configuration settings from a test environment to the production environment. For information about migrating projects, see Migrating projects from the test environment to the production environment (page 275).

**Before you can import the project configuration, you must first run the import_project_ repository command described in Importing a project (page 272).**

**Syntax**

`ComposeCli.exe import_project_repository_config --project project_name --infile input-file [--password password] [--is_without_credentials]`

where:
• **project** is the name of the project you want to export.

• **infile** specifies the path to and name of the input file. This file is in JSON format. For example: C:\file.json.

• **is_without_credentials** specifies to import the project configuration without the encrypted fields. In this case, you will need to manually enter the project’s Landing Zone and Storage Zone passwords (in the Data Zone Connection settings).

• **password** the password specified with the `password` qualifier when the project configuration was exported.

For instructions on changing the Master User Password, see *Changing the master user password (page 31).*

See also: *Migrating projects from the test environment to the production environment (page 275)* and *Import/Export scenarios: When is a password required? (page 276)*

### Migrating projects from the test environment to the production environment

After successfully creating and testing projects in the test environment, you now want to move those projects to the production environment. You also need to propagate updates from the testing environment to the production environment as necessary. Although it sounds complicated, moving new and updated projects from the test environment to the production environment is actually quite straightforward, as explained below.

See also *Import/Export scenarios: When is a password required? (page 276).*

> **Landing and Storage Connections (landing, storage and provisioning) will not be overridden when moving to a production environment. This also includes the file format set in the provisioning task.**

> **The Landing Zone and Storage Zone display names must be identical in both the testing and the production environments.**

To perform the initial migration from the testing environment to the production environment:

1. Export the project from the test environment as described in *Exporting a project (page 271).*
2. Import the test project to the production environment as described in *Importing a project (page 272).*
3. Edit the connection settings to point to the production Landing Zone and Storage Zone.

For more information, see *Defining Landing Zones (page 292)* and *Defining a connection to the Storage Zone (page 284)* respectively.

4. Configure notifications and scheduling as needed.

For more information, see *Scheduling tasks (page 334)* and *Notifications (page 335)* respectively.

To propagate updates from the testing environment to the production environment:

1. Export the project from the test environment as described in *Exporting a project (page 271).*
2. Import the test project to the production environment as described in *Importing a project (page 272).*
Import/Export scenarios: When is a password required?

The following section describes which of the various export/import scenarios require a password to be specified.

In all scenarios, if you import a project to an existing project, the credentials of the existing projects are preserved (as they are part of the project configuration).

**Scenario 1:** Moving a project or project configuration between two Compose machines without retaining the project credentials. This is useful when importing to a new project that will have different project credentials.

In such a scenario, simply add the `is_without_credentials` parameter to either the export or the import command.

**Scenario 2:** Moving a project or project configuration between two Compose machines that have the same Master User Password.

In such a scenario, neither the export command nor the import command need to include a password. If you do not want the source and target projects to have the same credentials (for Data Zone connectivity, etc.), then you also need to specify the `is_without_credentials` parameter in either the export or the import command.

**Scenario 3:** Moving a project or project configuration between two Compose machines that have a different Master User Password, but without revealing the Master User Password of the source machine.

In such a scenario, the export command must include the `password` and `master_user_password` parameters while the import command must include the `password` parameter. The same password (specified with the `password` parameter) must be used for both export and import.

**Scenario 4:** Moving a project or project configuration between two Compose machines that have a different Master User Password.

In such a scenario, the export command does not need to include a password, but the import command should specify the Master User Password of the source machine (using the `password` parameter).

Generating projects using the CLI

The instruction below explain how to automatically generate projects using the CLI. This can be especially useful when deploying projects between different environments.

On the Compose machine:

1. From the **Start** menu, open the **Compose Command Line** console.
2. Run the following command:
   ```
   ComposeCli.exe connect
   ```
3. Run the following command:
   ```
   ComposeCli.exe generate_project --project project_name [--database_already_adjusted]
   ```
   Where:
project_name is the name of your project.
--database_already_adjusted is an optional parameter that should only be included if the storage was adjusted outside of Compose.

When the command is run, Compose will:

- Validate the metadata.
- Create any storage tables that do not exist.
- Validate the storage.
- Adjust the storage if needed.

If an Adjust script is needed and --database_already_adjusted is included in the command, the script (DDL) will not be run as it is assumed that the user ran it manually.
- If the "Adjust" cannot be performed automatically, the process will be stopped.

- Generate all storage tasks.

If Compose encounters an error while generating a storage task, it will skip the problematic task and continue with the remaining tasks.

Viewing and downloading DDL scripts

In the DDL Script Files window, you can view and download the Storage Zone DDL script files. By default, Compose executes the Create, Adjust and Drop statements immediately upon user request. However, when the Generate DDL scripts but do not run them option is enabled, Compose will only generate the scripts but not execute them.

For more information on the Create DDL scripts only option, see Project settings (page 263).

To open the DDL Script Files window:

1. Open your project as described in Managing and monitoring projects (page 262).
2. Click the downward arrow to the right of the project name and select Show DDL Scripts from the drop-down menu.

The DDL Script Files window opens.
3. To view a script, select the desired script in the **Script Files** pane on the left. The script will be displayed on the right.

4. To download a script, select the desired script in the **Script Files** pane on the left. Then click the download button in the top right of the window.

5. To search for an element in the script, start to type in the search box. All strings that match the search query will be highlighted blue.
   You can navigate between search query matches using the arrows to the right of the search box.
   Use the right and left single arrows to navigate matches sequentially. Use the right and left double arrows to jump to the last and first match respectively.

6. To reset the search, either delete the search query or click the "x" in the right of the search box.

**Project versioning**

Compose provides built-in project version control using the Git engine. Version control enables Compose developers to commit project revisions to both a local and a remote Git repository. If a mistake is made, Compose developers can easily roll back to earlier versions of the project while minimizing disruption to all team members.

*Revisions only store metadata and mapping information. After you revert to a saved revision, you will need to recreate the data warehouse and data mart tables.*
Configuring version control settings

To define Version Control Settings:

1. From the project drop-down menu, select Version Control > Settings. The Version Control Settings - Git window opens. The Local Commits area shows the local root folder where project revisions are committed. The first time a project revision is committed, Compose creates a JSON file with the current project settings. The <project_name>.json file is archived to a ZIP file (<project_name>_deployment.zip), which is located in a project-specific folder under the source-control folder.

2. To enable commits to a remote Git database, select Enable remote commits and then provide the following information:
   - URL - The address of the remote Git database.
   - User name - Your user name for accessing the remote Git database.
   - Password - Your password for accessing the remote Git database.

Committing projects

You can commit a project using the console or using the CLI:

To commit a project to Version Control using the web console:

1. From the project drop-down menu, select Version Control > Commit. The Commit - <Project_Name> window opens.
2. Enter a message in the Message box and optionally select the Remote push check box. Note that the Remote push check box will be disabled if the Enable remote commits option described above is not selected.

To commit a project to Version Control using the CLI:

Run the following command from the Compose bin directory:

```bash
composeCli.exe commit --project project_name [--message message] [--remote]
```

Where:

- commit is the verb.
- project_name is the name of the project you want to commit.
- message is an optional message to accompany the commit.
- remote is required if you want to commit the project to a remote Git repository (see above). By default, the project will be committed locally to <product_dir>/data/source-control.

To revert to a saved revision:

1. From the project drop-down menu, select Version Control > Revisions history. The Revision History - <Project_Name> window opens. By default, the last 10 revisions are shown. You can change this number by selecting one of the available options from the Show drop-down list.
2. Optionally, use the Search box to find a specific revision.
3. Select the desired revision and then click the Deploy to Revision toolbar button.
4. When prompted to confirm the operation, click Yes. The existing project will be replaced.
5. Click Close to close the Revision History - <Project_Name> window.

To download a saved revision:

1. From the project drop-down menu, select Version Control > Revisions history. The Revision History - <Project_Name> window opens. By default, the last 10 revisions are shown. You can change this number by selecting one of the available options from the Show drop-down list.
2. Optionally, use the Search box to find a specific revision.
3. Select the desired revision and then click the Download Revision as Package toolbar button. The package will be saved as a ZIP file in your browser’s default download location.

Creating a diagnostics package

To assist in troubleshooting esoteric issues, a Qlik Support Engineer may ask you for a diagnostics package. The diagnostics package contains the following information:

- The project "data" directory
- Java logs and workflow logs
- .NET logs
- Deployment package file

As a prerequisite to creating a diagnostics package, the project must have at least one database connection configured.

To create a diagnostics package:

1. From the Project menu, select Create Diagnostics Package.
2. A zip file in the following format will either be downloaded to your computer or you will be prompted to download it (according to your browser settings):
   Compose_Diagnostics_<project_name>_<timestamp>.zip

6.3 Getting started with Data Lake projects

This section provides an overview of the Data Lake project workflow, familiarizes you with the console elements and explains how to set up a task in Qlik Replicate.

In this section:

- High-level flow (page 281)
- Console elements (page 281)
High-level flow

A Qlik Compose workflow is typically set up as follows (simplified):

1. In Replicate, define a task that replicates the source tables to a specific target. The target should be defined as the Landing Zone in your Qlik Compose project.
2. In Compose:
   a. Configure access to your Storage Zone and your Landing Zone(s).
   b. Use the "Discover" option to auto-generate the metadata from the source tables located in the Landing Zone(s). You can even create the Metadata manually if you prefer.
   c. Optionally, create the Storage Zone tables and then generate the ETL statements that will be executed when the task runs.
   d. Run the separate Full Load and CDC tasks (in that order) that were automatically created when the source tables were discovered.

See also Introduction (page 15).

Console elements

This section will familiarize you with the elements that comprise the Qlik Compose UI.

To open Qlik Compose:

From the Windows Start menu, select All Programs > Qlik Compose > Qlik Compose Console.

The Qlik Compose Console opens in Management view.

Qlik Compose Console - Management View

Management View

In Management view, you can manage the following:

- Qlik Compose projects
  For more information, see Adding and managing data warehouse projects (page 35).
- The product license
- Replicate Server connections
6 Data Lake projects

- Compose Agent connection
- Log levels and cleanup options
- Email settings
- User permissions

For more information, see Managing Compose (page 343).

Designer View

When you add a new project or open an existing project, the console switches to Designer view. You can switch back and forth between Designer view and Monitor view by clicking the Designer and Monitor tabs in the top right of the console.

Designer view comprises the following panels:

- **Landing and Storage Connections** - Configure access to your Landing Zone(s) and Storage Zone. For more information, see Defining Landing Zones (page 292) and Defining a connection to the Storage Zone (page 284) respectively.
- **Storage Zone** - In the Storage Zone, you can:
  - Discover and manage the source table metadata.
  - Define data storage tasks that move the data from the Landing Zone(s) to the Storage Zone. For more information, see Selecting source tables and managing metadata (page 294) and Creating and Managing Storage Zone Tasks (page 315).

In Designer view, each of the panels has a bar below the panel name. The bar can be empty, half-filled or completely filled, according to the current configuration status of the panel properties, as follows:

No fill (gray) - Not configured

![No fill](image)

Partially filled - Configuration is not complete

![Partially filled](image)
Data Lake projects

Completely filled - Fully configured

Monitor View

To switch to Monitor view, click the Monitor tab in the top right of the console.

Monitor View

In Monitor view, you can view the status of Qlik Compose tasks, schedule their execution (either individually or as a workflow), view logs, and create notifications.

For more information, see Controlling and monitoring tasks and workflows (page 331).

6.4 Setting up landing and storage connections

This topic explains how to configure connectivity to your Storage Zone and Landing Zone(s).

In this section:
Defining a Storage Zone

This section explains how to set up Storage Zone connectivity in a Qlik Compose project.

In this section:

- Defining a connection to the Storage Zone (page 284)
- Supported data types (page 291)
- Required permissions (page 284)

Required permissions

The following permissions are required:

- **Metadata**: Read and Write
- **Tables**: Insert and Update, and Delete.

Defining a connection to the Storage Zone

As the server connection settings for the Landing Zone are derived from the Storage Zone settings, you must define a Storage Zone before defining a Landing Zone.

For more information on adding data sources, see Defining Landing Zones (page 292).

To define the Storage Zone connection:

1. Open your project and click the Manage button in the bottom left of the Databases panel.
   The Manage Databases window opens.
2. Either, click the Add New Database link in the middle of the window.
   -OR-
   Click the New toolbar button.
   The New Storage window opens. The settings will be relevant for the compute platform you selected when you set up your project. The sections below detail the settings according to each of the available compute platforms.

To use AVRO file format with Hive 3.x, you must set the following parameter:

```
metastore.storage.schema.reader.impl=org.apache.hadoop.hive.metastore.SerDeStorageSchemaReader
```
Cloudera Compute Platform

Security

Use SSL - Select to connect using SSL.

- **Use self-signed certificate** - Select to connect using a self-signed certificate.
- **Trusted store full path** - Enter the full path to the store containing your trusted certificates.
- **Trusted store password** - Enter the password for your trusted certificate store.

Authentication Type:

- **User name** - Select to connect to the Hadoop cluster with only a user name. Then, in the **User name** field, specify the name of a user authorized to access the Hadoop cluster.

- **User name and password** - Select to connect to the Hadoop cluster with a user name and password. Then, in the **User name** and **Password** fields, specify the name and password of a user authorized to access the Hadoop cluster.

- **Knox** - Select this option if you need to access the Hortonworks Hadoop distribution through a Knox Gateway. Then, provide the following information:
  - **Host** - The FQDN (Fully Qualified Domain Name) of the Knox Gateway host.
  - **Knox port** - The port number to use to access the host. The default is "8443".
  - **Knox Gateway path** - The context path for the gateway. The default is "gateway".

> The port and path values are set in the gateway-site.xml file. If you are unsure whether the default values have been changed, contact your IT department.

- **Cluster name** - The cluster name as configured in Knox. The default is "Default".
- **User name** - Enter your user name for accessing the Knox gateway.
- **Password** - Enter your password for accessing the Knox gateway.

- **Kerberos** - Select to authenticate against the Hadoop cluster using Kerberos. Then, provide the following information:
  - **Realm**: The name of the realm in which your Hadoop cluster resides. For example, if the full principal name is john.doe@EXAMPLE.COM, then EXAMPLE.COM is the realm.
  - **Principal**: The user name to use for authentication. The principal must be a member of the realm entered above. For example, if the full principal name is john.doe@EXAMPLE.COM, then john.doe is the principal.
  - **Keytab file**: The full path of the Keytab file. The Keytab file should contain the key of the Principal specified above.

> The krb5.ini file should be located in C:\Windows (according to the Java default). However, if Replicate is installed on the same machine as Compose, the file might be in C:\Program Files\MIT\Kerberos. In such a case, simply copy the file to C:\Windows.
• **Host:** The FQDN that will be used to locate the correct Principal in Kerberos. This is only required if the IP address of the Hive machine is not known to Kerberos.

• **Service name:** The default is "hive". You should only change this if you are sure that the service name is different.

   
   In case of an issue with the Kerberos authentication, do the following:
   1. Test the connection to the Hive machine with Kerberos.
   2. Check the Kerberos configuration on HDFS.

If you are unsure about any of the above, consult your IT administrator.

**Hive Access**

• **Use ZooKeeper** - Select if your Hive machines are managed by Apache ZooKeeper.

• **ZooKeeper hosts** - The machines that make up the ZooKeeper ensemble (cluster). These should be specified in the following format:
  
  host1:port1,host2:port2,host3:port3

• **ZooKeeper namespace** - The namespace on ZooKeeper under which the HiveServer2 znodes are located.

• **Host** - If you are not using ZooKeeper, specify the IP address of the Hive machine. This should be the same as the host name or IP address specified in the Cloudera Data Platform Private Cloud or Hortonworks Data Platform target endpoint settings in the Replicate task.

• **Port** - If you are not using ZooKeeper, optionally change the default port.

• **Database name** - Specify the name of the Hive target database. This must be different from the database specified in the Landing Zone settings.

   
   If the database does not exist Compose will try and create it. This requires the Compose user to be granted the necessary permission to create the database.

• **JDBC parameters** - Additional parameters to add to the default Simba JDBC connection string. These should be key values separated by a semi-colon.

   **Example:**
   
   KEY=VALUE;KEY1=VALUE1

   
   • You can set Hive parameters in the JDBC parameters. For example:
     
     - `mapred.job.queue.name=<queue_name>`
     - `hive.execution.engine=<engine_name>`

   • To distinguish Compose Hive sessions from other Hive Sessions, if Tez is being used, you can define a JDBC parameter to change the query description, as follows:
     
     `hive.query.name=my_description`
Amazon EMR Compute Platform

**Security**
- **Use SSL** - Select to connect using SSL.

**Authentication type:**
- **User name** - Select to connect to the Hadoop cluster with only a user name. Then, in the User name field, specify the name of a user authorized to access the Hadoop cluster.
- **User name and password** - Select to connect to the Hadoop cluster with a user name and password. Then, in the User name and Password fields, specify the name and password of a user authorized to access the Hadoop cluster.

If you are unsure about any of the above, consult your IT administrator.

**Hive Access**
- **Host** - Specify the IP address of the Hive machine. This should be the same as the host name or IP address specified in the Amazon EMR target endpoint settings in the Replicate task.
- **Port** - Optionally change the default port.
- **Database name** - Specify the name of the Hive target database. This must be different from the database specified in the Landing Zone settings.

*If the database does not exist Compose will try and create it. This requires the Compose user to be granted the necessary permission to create the database.*

- **JDBC parameters** - Additional parameters to add to the default Simba JDBC connection string. These should be key values separated by a semi-colon.

**Example:**
KEY=VALUE;KEY1=VALUE1

- You can set Hive parameters in the JDBC parameters. For example:
  - `mapred.job.queue.name=<queueName>`
  - `hive.execution.engine=<engineName>`

*To distinguish Compose Hive sessions from other Hive Sessions, if Tez is being used, you can define a JDBC parameter to change the query description, as follows:*

  `hive.query.name=my_description`

- **Hive metadata storage type** - Select one of the following storage mediums for your Hive metadata:
  - **Hive Metastore** - This is the default metadata storage type.
  - **AWS Glue Data Catalog** - You can choose to store Hive metadata using the AWS Glue Data Catalog. AWS Glue allows you to store and share metadata in the AWS Cloud in the same way as in a Hive metastore.
When using AWS Glue Data Catalog for metadata storage, Compose control tables will be created with the data type STRING instead of VARCHAR (LENGTH).

Databricks Compute Platform

Security

All connections to Databricks use SSL.

- **Authentication type** - "Databricks Delta". This cannot be changed.
- **User name** - The default user name is "token", which requires you to specify your personal access token in the Password field. Although it's strongly recommended to use a token, you can also access your Databricks account using a standard user name and password.
- **Password** - If you are using a personal access token, this will be the token value. Otherwise, specify the password for accessing your Databricks account.
- **HTTP Path** - The HTTP path to your Databricks compute resources.
  
  **Example:**
  
  sql/protocolv1/o/8388045294310983/0123-xxxxxx-xxxxxxx

If you are unsure about any of the above, consult your IT administrator.

Hive Access

- **Host** - Specify the IP address of the Hive machine. This should be the same as the host name or IP address specified in the Databricks target endpoint settings in the Replicate task.
- **Port** - Optionally change the default port.
- **Database name** - Specify the name of the Hive target database. This must be different from the database specified in the Landing Zone settings.

  - **If the database does not exist Compose will try and create it. This requires the Compose user to be granted the necessary permission to create the database.**
  - **To prevent table name conflicts when using Databricks, the Landing Zone and Storage Zone databases should be different.**

- **JDBC parameters** - Additional parameters to add to the default Simba JDBC connection string. These should be key values separated by a semi-colon.

  **Example:**
  
  KEY=VALUE;KEY1=VALUE1
You can set Hive parameters in the JDBC parameters. For example:

- `mapred.job.queue.name=<queuename>`
- `hive.execution.engine=<enginename>`

To distinguish Compose Hive sessions from other Hive Sessions, if Tez is being used, you can define a JDBC parameter to change the query description, as follows:

- `hive.query.name=my_description`

HDInsight Compute Platform

Security

- **User name** - Specify the name of a user authorized to access the Hadoop cluster.
- **Password** - Specify the password of the user specified in the **User name** field.

Hive Access

- **Host** - Specify the IP address of the Hive machine. This should be the same as the host name or IP address specified in the Microsoft Azure HDInsight target endpoint settings in the Replicate task.
- **Port** - Optionally change the default port.
- **Database name** - Specify the name of the Hive target database. This must be different from the database specified in the Landing Zone settings.

If the database does not exist Compose will try and create it. This requires the Compose user to be granted the necessary permission to create the database.

- **JDBC parameters** - Additional parameters to add to the default Simba JDBC connection string. These should be key values separated by a semi-colon.

**Example:**

```
KEY=VALUE;KEY1=VALUE1
```

You can set Hive parameters in the JDBC parameters. For example:

- `mapred.job.queue.name=<queuename>`
- `hive.execution.engine=<enginename>`

To distinguish Compose Hive sessions from other Hive Sessions, if Tez is being used, you can define a JDBC parameter to change the query description, as follows:

- `hive.query.name=my_description`
Dataproc Compute Platform

Security

- **Use SSL** - Select to connect using SSL.

Authentication type:

- **User name** - Select to connect to the Hadoop cluster with only a user name. Then, in the **User name** field, specify the name of a user authorized to access the Hadoop cluster.
- **User name and password** - Select to connect to the Hadoop cluster with a user name and password. Then, in the **User name** and **Password** fields, specify the name and password of a user authorized to access the Hadoop cluster.

If you are unsure about any of the above, consult your IT administrator.

Hive Access

- **Host** - Specify the IP address of the Hive machine. This should be the same as the host name or IP address specified in the Google Dataproc target endpoint settings in the Replicate task.
- **Port** - Optionally change the default port.
- **Database name** - Specify the name of the Hive target database. This must be different from the database specified in the Landing Zone settings.

*If the database does not exist Compose will try and create it. This requires the Compose user to be granted the necessary permission to create the database.*

- **JDBC parameters** - Additional parameters to add to the default Simba JDBC connection string. These should be key values separated by a semi-colon.

  **Example:**
  
  KEY=VALUE;KEY1=VALUE1

  *You can set Hive parameters in the JDBC parameters. For example:*
  
  - `mapred.job.queue.name=<queueName>`
  - `hive.execution.engine=<engineName>`

  *To distinguish Compose Hive sessions from other Hive Sessions, if Tez is being used, you can define a JDBC parameter to change the query description, as follows:*

  `hive.query.name=my_description`

After defining your Storage Zone connection parameters:

1. **Click Test Connection** to verify that Compose is able to establish a connection with the specified database.
2. **Click OK** to save your settings. The database is added to the list on the left side of the **Manage Databases** window.
Supported data types
The following table shows the default mapping from Compose data types to Apache Hive and Databricks data types.

<table>
<thead>
<tr>
<th>Compose Data Types</th>
<th>Hive Data Types</th>
<th>Databricks Data Types</th>
</tr>
</thead>
<tbody>
<tr>
<td>INTEGER</td>
<td>INT</td>
<td>INT</td>
</tr>
<tr>
<td>DATETIME</td>
<td>TIMESTAMP</td>
<td>TIMESTAMP</td>
</tr>
<tr>
<td>TIME</td>
<td>TIMESTAMP</td>
<td>TIMESTAMP</td>
</tr>
<tr>
<td>DATE</td>
<td>DATE</td>
<td>DATE</td>
</tr>
<tr>
<td>BIGINT</td>
<td>REAL</td>
<td>BIGINT</td>
</tr>
<tr>
<td>BYTE ARRAY</td>
<td>STRING</td>
<td>STRING</td>
</tr>
<tr>
<td>DECIMAL</td>
<td>DECIMAL (P,S)</td>
<td>DECIMAL (P,S)</td>
</tr>
<tr>
<td>GUID</td>
<td>VARCHAR (38)</td>
<td>STRING</td>
</tr>
<tr>
<td>VARCHAR</td>
<td>VARCHAR (LENGTH)</td>
<td>VARCHAR (LENGTH)</td>
</tr>
<tr>
<td>STRING</td>
<td>STRING</td>
<td>STRING</td>
</tr>
</tbody>
</table>

Working with views
Compose creates the Storage Zone with both storage tables and storage views. The storage tables are created in the database that you defined in your storage settings while two separate database are created for the views: The exposed views database and the internal views database. The exposed views database is the primary views database and contains all view types. The internal views database is used to store updates to ODS Live Views and HDS Live Views. The exposed views database and the internal views database share the same name as the Storage Zone database, but are appended with a unique suffix (by default, _v and _v_internal respectively), which is set in the project settings' Naming tab. Consuming applications should be set up to read from the exposed views database, which provide several benefits over tables including better security (requiring read-only access only), data concurrency, and minimizing duplicate records in projects defined with non-ACID storage.

There are four types of view, depending on the project-level or entity-level data store type:

- **ODS standard views** - Created when the data store type is ODS only. These views will always reflect the same data unless the storage task is run.
- **ODS Live Views** - Created when the data store type is ODS only. As opposed to standard views, these views always reflect any changes to the Replicate Change Tables and to the Storage tables.
- **HDS standard views** - Created when the data store type is ODS + HDS. These views contain both current records and historical records and will only be updated if you run storage tasks.
- **HDS Live Views** - Created when the data store type is ODS + HDS. These views contain both current records and historical records. As opposed to standard views, these views always reflect any changes to the Replicate Change Tables and to the Storage tables.
When using live views, to ensure transactional consistency, it is recommended to turn off Speed partition mode in the Replicate task settings. When set to off, Replicate will close the partition only at the end of each transaction. This might require you to shorten the partition interval in order for the changes to be propagated to Compose in a timely manner. Shortening the partition interval might also require you to increase the partition cleanup frequency to prevent too many files from accumulating on the target and degrading performance.

For information about turning off Speed partition mode, setting partitioning intervals, and partition cleanup, see the Replicate Help.

Tables that were reloaded in Replicate will be automatically reloaded in Compose the next time the task runs. To prevent data inconsistency, Live Views should not be read while the tables are being reloaded.

Standard views contain data that was already applied to the storage tables, with mid to low-level latency. As consuming data from standard views requires less computing resources, this should be the first choice for downstream users. However, if latency is too high for some applications, Live views can be used instead. Although using live views significantly reduces latency, doing so requires greater computational resources. There is also the possibility that the date in live views might be less consistent than the data in standard views as updates may not have been applied to all the storage tables at the same time.

Although the views are in a separate database, you can use the suffixes (specified in the project settings’ Naming tab) to help identify them.

Defining Landing Zones

This section explains how to set up Landing Zone connectivity in a Qlik Compose project.

In this section:

- Landing Zone permissions (page 292)
- Defining Landing Zones connections (page 292)

Landing Zone permissions

For proper operation, the Landing Zone database must be granted the following permissions:

- Read metadata
- Select from tables

For information on configuring the Landing Zone, see Defining Landing Zones connections (page 292).

Defining Landing Zones connections

In a Data Lake project, you can define any number of Landing Zone connections. Defining multiple Landing Zone connections is necessary if the data that you eventually want to be available in your Storage Zone is located in several Landing Zones.
The Landing Zone connection settings tell Compose where the source tables from the Replicate task are located. Since the Landing Zone is always located on the Storage Zone Server and the Storage Zone connection details have already been defined, you do not need to provide them again.

- **Before you can define a Landing Zone connection in a Data Lake project, you first need to define a Storage Zone connection.**

For more information on defining a Storage Zone connection, see *Defining a connection to the Storage Zone (page 284).*

To define a Landing Zone connection:

1. Open your project and click **Manage** in the **Databases** panel. The **Manage Databases** window opens.
2. Click the **New** toolbar button. The **New Data Source** window opens.
3. In the **Name** field, specify a display name for your Landing Zone.
4. From the **Content type** drop-down list, choose whether the content in the landing zone is **Full Load Only, Change Processing** or **Full Load and Change Processing** (according to the Qlik Replicate task definition).
5. In the **Database name** field, specify the database name. This must be the same as the Hive access database defined in the Qlik Replicate target endpoint (in the Qlik Replicate task).

   - **To prevent table name conflicts, the Landing Zone and Storage Zone databases should be different.**

   For more information, see *Defining a Qlik Replicate task (page 34).*

6. **Associate with Replicate Task** - This is required. Select this to associate your Data Lake project with the related Replicate task. Replicate tasks replicate the relevant tables from the source database to the Landing Zone. Specifying the Replicate task name will enable you to monitor and control that task from within Compose.

   - **Before you can choose a Replicate task however, you first need to define the connection settings to the Qlik Replicate Server machine. To do this, click the Replicate Server Settings link below the Task field and then configure the settings as described in Replicate Server settings (page 347).**

Once you have configured connectivity to at least one Replicate Server, you can then proceed to select a Replicate task.

To select a Replicate task:

1. Click the browse button to the right of the **Associate with Replicate task** field. The **Select Replicate Task** window opens.
2. Select a Replicate Server from the **Server** drop-down list.
The Replicate Tasks list is populated with all tasks defined on the selected server.

3. Select the task that is replicating the source tables to the landing zone and then click OK.

The name of the selected task is shown as read-only in the Associate with Replicate task field.

7. Click Test Connection and then, if the connection is successful, click OK to save your settings.

Managing Landing and Storage connections

You can edit and delete Landing and Storage Connections as required. The table below describes the available options.

<table>
<thead>
<tr>
<th>To</th>
<th>Do this</th>
</tr>
</thead>
<tbody>
<tr>
<td>Edit a Data Zone connection</td>
<td>In the left side of the Manage Landing and Storage Connections window, select the desired Data Zone (Landing Zone or Storage Zone) and then click the Edit toolbar button.</td>
</tr>
<tr>
<td>Delete a Data Zone connection</td>
<td>In the left side of the Manage Landing and Storage Connections window, select the desired Data Zone (Landing Zone or Storage Zone) and then click the Delete toolbar button. Click Yes when prompted to confirm the deletion.</td>
</tr>
</tbody>
</table>

6.5 Selecting source tables and managing metadata

This section describes how to select source tables and manage metadata. The source tables are the tables that were replicated to the Landing Zone by the Replicate task (i.e. the target tables of the Replicate task).

In this section:

- Selecting and adding the source tables (page 294)
- Validating the metadata and storage (page 298)
- Managing the metadata (page 300)
- Creating transformations (page 307)
- Reusable transformations (page 313)

Selecting and adding the source tables

This section explains how to select and add the source tables. Note that in the following explanations, "table" refers to the physical database object whereas "entity" refers to the logical object within Compose.

In this section:

- Discovering the Landing Zone (page 295)
- Importing entities and mappings from another project (page 296)
- Clearing the metadata cache (page 328)

See also Managing entities (page 300) for information on adding entities manually.
Discovering the Landing Zone

If you want the metadata to be created with Primary Keys, you need to associate a Replicate task with the Landing Zone. For instructions on how to do this, see Defining Landing Zones connections (page 292).

To discover the source tables:

1. Open your project.
2. In the Metadata panel, select Discover from the drop-down menu in the top right corner.
   -OR-
   In the Manage Metadata window, click the Discover toolbar button on the left.
   The Discover window opens.
3. Select the desired source Landing Zone and then click OK.
   The Source Tables/Views Selection - Name window opens.
4. Choose one of the following Search for options:
   a. To search for tables only, select Tables.
   b. To search for views only, select Views.
   c. To search for tables and views, select All.
5. To include internal Qlik tables in the search results, select the Show Internal Qlik Tables check box. This may be useful for debugging, but is not usually not necessary.
6. To only search for tables/ views whose names contain a specific string, type the string in the Name field.
   For example, entering "ers" will return "customers" and "suppliers" in the search results.
7. Click the Search button.
   The resulting tables/views will be displayed in the list in the left of the window.
8. Optionally, click the Clear Cache button to clear the Landing Zone's metadata cache (tables and columns). This may be necessary, for example, if tables were added to the Landing Zone or renamed. Such tables will not appear in the table list until the cache is cleared.
9. To add all of the resulting tables/views, click the >> button (Add All)
   You can select multiple tables/views by holding down the [Shift] (sequential selection) or [Ctrl] (non-sequential selection) button.
10. To add specific tables/views, select the desired tables and/or views and then click the > (Add) button.
If you add a table that already exists in the Metadata with the same name, then the new table is added with the name: source_table_name_DISCOVERED (or source_table_name_DISCOVERED_02 if the name source_table_name_DISCOVERED already exists, and so on).
If the table contains attribute domains that differ from existing ones but have the same name, they will also be appended with the _01 suffix.

11. Click OK to add the selected tables/views to the project.
The Generating Metadata from [Metadata Name] window opens.
A progress bar indicates the current metadata generation progress. For each stage of the metadata generation process, a corresponding message appears in the Messages list.

12. After the metadata has been generated, click Close.
13. Repeat steps 2-12 to discover additional sources.

Importing entities and mappings from another project
You can import entities and mappings from another project with the same Storage Zone type. This can be useful within a development environment, for example, if you need to integrate a private developer's project with the main project.

To import entities and mappings
1. Open the Manage Metadata window as described in Managing the metadata (page 300).
2. In the Entities toolbar, click the Import from Project button.
3. The Import from Project wizard opens.
4. In the Entities tab:
   - Select a project from the Import from Project drop-down list.
   - Optionally, search for specific entities.
   - Select which entities to import or select Select All to import all entities.
5. Click Next to select which mappings to import.

To create new entities and mappings if the selected entities and mappings already exist, clear the Replace existing entities and mappings check box.
The new entities/mappings will be named <existing_name>_IMPORTED (or <existing_name>_IMPORTED_<n> if the entity/mapping is imported more than once).

6. In the Mappings tab:
   - Either click Finish to import all mappings for the selected entities (the default).
   - OR-
   - Select which mappings you want to import and then click Finish to import the selected entities and mappings.
Clearing the metadata cache
To improve performance when reading from the Landing Zone or from the Storage Zone, Compose caches both the Landing Zone metadata and the Storage Zone metadata. However, synchronization issues may sometimes occur if the structure of the Landing Zone or the Storage Zone metadata is altered outside of the Compose project.

If you are aware of external changes to the metadata or if you notice any data synchronization anomalies, Compose enables you to clear the metadata cache, either using the web console or using the CLI.

Clearing the landing zone metadata cache
To clear the landing zone metadata cache and refresh the mappings on the next reading of the metadata:

1. Click the Manage button at the bottom left of the Storage Zone panel.
2. Click the Clear Landing Cache button in the Manage Storage Tasks window.

See also the section describing how to clear the cache before discover.

Clearing the storage zone metadata cache
To clear the storage zone metadata cache:

1. In the Storage Zone panel, select the Clear Metadata Cache item from the menu in the top right corner.
2. Click Yes to clear the storage zone metadata.
3. When the storage zone metadata cache has been successfully cleared, click Close.

Clearing the metadata cache using the CLI

To clear the metadata cache using the CLI:

1. From the Start menu, open the Compose Command Line console.
2. Run the following command:
   ```
   ComposeCli.exe connect
   ```
3. Run the following command:
   ```
   ComposeCli.exe clear_cache --project project_name [--type landing|storage] [--landing_zone source_name]
   ```

   Where:
   - `--project` is the name of your project.
   - `--type` is where to clear the metadata cache. Possible values are landing or storage. If `--type landing` and you want to clear a specific landing zone, you must set the `--landing_zone` parameter as well. To clear the metadata cache in all landing zones, specify
--type landing and omit the --landing_zone parameter.
• --landing_zone is the name of the landing zone when --type landing_zone.

Example:

ComposeCli.exe clear_cache --project myproject --type landing --landing_zone mysourcel

Validating the metadata and storage

Once the table metadata has been generated, to prevent data inconsistency issues, it is strongly recommended to check the validity of the metadata and the Storage Zone. For example, for the metadata to be valid, each of the tables must have a Business Key.

Validating the metadata does not recalculate expressions for historical data that has changed.

To validate the metadata:

1. Click the Validate button in the bottom right of the Metadata panel. Compose will run validation checks and identify any entities which are not valid. If the metadata is valid, the following message will be displayed:
   Validation tests completed successfully. No issues were detected.
   If the metadata is not valid, the Validating Storage Zone window opens. This window is divided into the following columns:
   • Severity: Warning or Error.
   • Message: A message indicating why the entity is invalid.
   • Names: The names of the affected entities.
   • Resolve: To open the Manage Metadata window and manually resolve the issue, click the Edit Entities button.

2. Resolve the issue (for example, by adding a Business Key) and then click Close. The Validate Metadata window will open.
3. Click the Refresh button in the top left corner. A message will confirm the metadata’s validity.
4. Click Close to exit the window.

To validate the Storage Zone:

1. Click the Validate button in the bottom right of the Storage Zone panel, or select Validate from the drop-down menu in the top right of the Storage Zone panel.

2. Compose will run a series of validation checks and the Validating the Storage window opens. If the Storage Zone metadata is not valid, the following message will be displayed:
   The metadata is not valid.
   If the Storage Zone is valid, the following message will be displayed:
   The Storage Zone is valid.
   If the metadata in the Manage Metadata window is not the same as the Storage Zone metadata, the following message will be displayed:
The Storage Zone is different from the metadata.

3. This step is only applicable if the Storage Zone metadata differs from the metadata in Compose. Review the report in the Metadata and Storage Comparison Report window and then do one of the following:

   • If all the changes can be adjusted automatically, do one of the following according to your configuration:
     • Click Adjust Automatically. The Adjust Storage Zone progress window opens. When the "The Storage Zone was adjusted successfully." message is displayed, close the window.
     • If the Generate DDL scripts but do not run them option is set, click Generate Adjust Script.

   ![Automatic adjust supports ADD ENTITY, DROP ENTITY, and ADD ATTRIBUTE (if it's the last attribute in the entity) only.]

   • If some or none of the changes cannot be adjusted automatically, the Adjust Automatically button will not be shown. In this case, do one of the following according to your configuration:
     • Click Drop and Recreate Tables. You will be prompted to confirm this action. Click Yes. The Dropping Storage Tables window opens. When the "The Storage Zone tables were dropped and recreated successfully." message is shown, close the window.
     • If the Generate DDL scripts but do not run them option is set, click Generate DDL Script.

Clicking Generate Adjust Script or Generate DDL Script

When you click Generate Adjust Script or Generate DDL Script, the Generate DDL Scripts window opens showing the progress of the script generation. The generated scripts will be saved to:

   `<product_dir>/data/projects/<project_name>/ddl-scripts`

Once the script(s) have been generated, close the Generate DDL Scripts window.

When working with a Hive-based compute platform, after you close the Generate DDL Scripts window, the DDL Script Files window opens automatically displaying the generated scripts. The DDL Script Files window provides a read-only view that allows you to review the scripts and download them.

The scripts need to be executed directly in your Storage Zone. Make sure that any modifications that you make to the scripts are done prior to executing them.

![When you run the adjust scripts, backup tables are created from the existing tables. The backup table names are appended with an "_old" suffix and must be deleted manually after the script completes.]

Setup and User Guide - Qlik Compose, August 2021 299
Search for "TODO" in the script to locate the part of the script that needs modifying.


See also: Supported characters (page 370).

Managing the metadata

You can add, remove and edit the entities and attributes according to your needs. All management tasks are performed in the Manage Metadata window, which you can open using one of the following methods:

- Click the Manage button at the bottom left of the Metadata panel.
- Click the Entities number in the Metadata panel.
- Select Manage from the drop-down menu in the top right of the Metadata panel.

The Manage Metadata window is split into two tabs: the Logical Metadata tab and the Physical Metadata tab. The Logical Metadata tab shows the entities and attributes as they appear in the Metadata whereas the Physical Metadata tab provides a preview of the actual tables (and columns) that will be created in the Storage Zone.

In the Logical Metadata tab, you can perform various management tasks such as adding and/or editing entities and attributes.

In this section:

- Managing entities (page 300)
- Managing attributes (page 301)
- Managing the Attributes Domain (page 303)

Managing entities

You can add, edit and remove entities as described in the table below.

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
</table>
| Add an entity | 1. Click the New Entity button in the Entities toolbar.  
2. Provide a name and description (optional) for the entity and then click OK.                                      |
| Edit an entity | 1. Select the entity you want to edit and then select Edit from the drop-down menu in the Entities toolbar.  
2. Edit the entity’s name and description (optional) and then click OK.                                                  |

Reducing the window size also shortens the toolbar. If the toolbar is too short to contain all the buttons, the toolbar options will be displayed in the drop-down menu instead. The shorter the toolbar, the more options will appear in the drop-down menu.
### To | Do This
--- | ---
Remove an entity | 1. Select the entity (or multiple entities) that you want to remove, and then select **Delete** from the drop-down menu in the **Entities** toolbar.
2. When prompted to confirm the deletion, click **Yes**.

Duplicate an entity | 1. Select the entity you want to duplicate and then select **Duplicate** from the drop-down menu in the **Entities** toolbar.
2. Edit the entity’s name and description (optional) and then click **OK**.
The duplicated entity is added to the Entities list.

Import entities from another project | See *Importing entities and mappings from another project* (page 296).

Include historical records | Select the check box in the **Save History** column to the right of the desired entity. Note that if you chose ODS +HDS as your data store, all of the **Save History** check boxes will be selected by default.

### Managing attributes
You can add, edit and remove attributes as required. All attributes in the Metadata belong to the Attributes Domain. When adding a new attribute, you can either select an existing attribute from the Attributes Domain or create a new Attributes Domain. Both of these options are described below.

To add an attribute from the attributes domain:

1. Click the **New Attribute** button in the **Attributes** toolbar.
The **New Attribute** window opens.
2. From the **Attribute domain** drop-down list, select the desired attribute.
3. To edit the selected attribute domain on-the-fly, click the edit button located after the **Attribute domain** drop-down list. This will open the **Edit - AttributeDomainName** window. Then, continue from Step 2 in **Edit an attribute domain**.
4. In the **Attribute name** field, optionally change the default instance name for the attribute domain.

*The name cannot contain any of the following forbidden (by Hive) characters:*

```
: ; . , ' "
```

You can create multiple instances of a single Attribute Domain. This is especially useful if you want to use the same Attribute Domain across multiple tables, with each "instance" having its own unique name. This also allows you to edit the properties of each attribute without affecting the other attributes, even though all of the Attribute Domain instances share the same parent Attribute Domain. For example, if the Attribute Domain name is "ID", you could create one instance for it in the "Categories" entity named "CategoryID" and another instance in the "Employees" entity named "EmployeeID". If, however, you edit the parent Attribute Domain attribute, all instances of that attribute will be updated as well.
5. **Data type:** The data type of the Attribute Domain. This can only be edited by editing the Attribute Domain.

6. To add a prefix to the attribute name, enter the desired prefix in the **Prefix** field. Adding a prefix to an attribute name allows you to add multiple instances of the same attribute domain. For example, the attribute "Employee" could become two different attributes: "ReportsTo_Employee" and "HiredBy_Employee".

7. To create an expression for the attribute, click the **fx** button located after the **Expression** field and then continue from **Creating transformations (page 307)**.

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

To create a new attribute domain and add it to the Metadata:

1. Click the **New Attribute** button in the Attributes toolbar. The **New Attribute** window opens.
2. Click the plus sign to the right of the **Attribute domain** drop-down list. The **New Attribute Domain** window opens.
   a. Specify a **Name** for the attributes domain.

   ```
   The name cannot contain any of the following forbidden (by Hive) characters:
   : / . , ' " :
   ```

   b. From the **Type** drop-down list, select one of the available data types.
   c. If the selected data type requires further configuration, additional fields will be displayed. For example, when Decimal is selected, the **Length** and **Scale** fields will be displayed. Set the values as desired.
   d. Optionally, specify a **Description**.
   e. Click **OK** to add the newly created attribute domain to the **Attribute domain** field and close the **New Attribute Domain** window.

3. Continue from **Step 4 in Add an existing attribute domain** above.

   ```
   You can also add new attribute domains via the Manage Attribute Domains window. For more information, see Managing the Attributes Domain (page 303)
   ```

To edit an attribute:

**Method 1:**

1. Select the attribute you want to edit and then click the **Edit** button in the Attributes toolbar. The **Edit Attribute Name** window opens.
2. Edit the values as required and then click **OK**.

**Method 2:**

1. Double-click the attribute you want to edit. The values in the attribute row become editable.
2. Edit the values as required and then click the tick button at the end of the row.

To remove an attribute:

1. Select the attribute(s) you want to delete.
2. Click the **Delete** button in the **Attributes** toolbar.
3. When prompted to confirm the deletion, click **Yes**.

```
The Storage Zone needs to be "adjusted" when deleting an attribute from the metadata and then adding the same attribute back to the metadata. However, the "Adjust" operation will also delete the data from the corresponding Storage Zone column.
```

To change the attribute order:

- Select the attribute you want to move and use the **Move Up/Move to Top** and **Move Down /Move to Bottom** toolbar buttons to move the attribute.

To manage the Attributes Domain:

- See *Managing the Attributes Domain* (page 303).

To create an expression for an attribute:

- See *Add an attribute from the attributes domain* or *Edit an attribute* above.

To export the attributes to a TSV file:

- Select an entity from the Entities list on the left of the **Manage Metadata** window and then select **Export to TSV** from the drop-down menu in the Attributes toolbar. Depending on your browser settings, you will either be prompted to download the `<entityname>_.Attributes.tsv` file or it will be downloaded to your default Downloads location.

**Managing the Attributes Domain**

The Attributes Domain provides a list of all the attributes available in the Compose metadata, as well as their data type. You can add, edit and delete attributes according to your data warehousing needs. The Attributes Domain also allows you to see which entities each attribute belongs to, as a single attribute may be present in several entities.

To manage the Attributes Domain:

1. From the drop-down menu in the top right of the **Storage Zone** panel, select **Manage Attributes Domain**.
2. Add, delete and edit attributes as describe in the table below.
To | Do This
--- | ---
Add an attributes domain | 1. Click the **New Attributes Domain** toolbar button. The **New Attribute Domain** window opens.
2. In the **Name** field, specify a name for the attribute.
   
   The name cannot contain any of the following forbidden (by Hive) characters:
   
   `; : , ' "`:
3. From the **Type** drop-down list, select one of the available data types.
4. If the selected data type requires further configuration, additional fields will be displayed. For example, when **Decimal** is selected, the **Length** and **Scale** fields will be displayed. Set the values as desired.
5. Optionally specify a **Description**.
6. Click **OK** to add the attribute and close the **New Attribute Domain** window.

   **Attribute domains names are case insensitive. For example, a project cannot contain one attribute domain called date and another called DATE.**

Edit an attribute domain | 1. Select the desired attribute and then click the **Edit** toolbar button. The **Edit: Name** window opens.
2. Edit the attribute as described in steps 2-6 of **Add an attributes domain** above.
   
   Note that the **Edit: Name** window also contains a **Used in Entities** list. Knowing which entities the attribute is used in may affect the type of changes you make, as the planned changes may not be appropriate for all entities.

Remove an attribute | 1. Select the attribute you want to delete and then click the **Delete** toolbar button.
2. When prompted to confirm the deletion, click **Yes**.

Schema evolution

Schema evolution allows users to easily detect structural changes to multiple data sources and then control how those changes will be applied to your project. Schema evolution can be used to detect all DDL changes that were made to the source database, although not all changes can be applied automatically (see “Supported data changes” below). Schema evolution can be performed using the web console or using the CLI as described below.

**Required permission:** Compose Designer or Admin.

Schema evolution requires certain options to be turned on in the Replicate task(s). For information about which options need to be enabled in Replicate for schema evolution, see *Defining a Qlik Replicate task* (page 255).
Supported data source changes

The following changes are supported:

- **New columns** - When this change is applied:
  - The new column will be added (as an attribute) to the end of the entity in the metadata
  - The column will be added to any mappings for that entity
  - The data warehouse will be adjusted
  - The storage task will be generated (but not run)

- **New tables** - When this change is applied:
  - The new table will be added (as an entity) to the metadata
  - A default mapping will be created
  - In the case of multiple storage tasks, the mapping will be added to the first task according to alphabetical order
  - The data warehouse will be adjusted
  - The storage task will be generated (but not run)

---

> Data warehouse adjustment and storage task generation will only occur if the appropriate apply option (see Step 5 below) was selected.

---

Schema evolution using the web console

1. Open the **Schema Evolution** window using one of the following methods:
   - Click the **Schema Evolution** button at the bottom of the METADATA panel.
   - Select **Schema Evolution** from the hamburger menu in the top right of the METADATA panel.
   - Open the **Manage Metadata** window and click the **Schema Evolution** toolbar button in the Logical Metadata tab.

2. Select which data sources to scan for changes. You can select either **All data sources** or **Selected data sources**. If you choose the latter, select which data sources to scan for changes.

3. Click **Scan for Changes**.
   
   During the scan, if a change is detected, the following message will be displayed:
   
   Found new DDL changes for data source <name>

   If no changes were detected, the following message will be displayed:
   
   No new DDL changes were found for data source <name>

   If unsupported changes were detected, one of the following messages will be displayed:
   
   DDL type <type> is not supported and will be skipped

   Column operation <operation> in data source <name> is not supported and will be skipped

4. When the scan completes, click **Close**.
   
   If changes were detected, the **Schema Changes** window opens showing the list of changes since the last scan (via the web console or the CLI).
5. Click **Apply Changes Options** if you want to apply the changes or **Ignore Changes** to close the window without applying the changes.

When you click **Apply Changes Options**, the **Apply Schema Changes** window opens, showing the following apply options:

- **Apply changes to the metadata and the mappings** - Select this option if you have other changes you would like to make before adjusting the data warehouse. You may even wish to undo some of the schema evolution changes as Compose does not allow you to choose which changes to apply.

- **Apply changes to the metadata and the mappings, and adjust the data warehouse** - Select this option if you have other changes you would like to make before generating the tasks.

- **Apply changes to the metadata and the mappings, adjust the data warehouse, and generate relevant tasks** - Select this option if you want to apply all the changes and generate associated tasks without making any prior changes.

6. Choose one of the options and click **Apply Changes**. The **Applying Schema Changes** window opens.

7. After the operations you selected in Step 5 have completed, click **Close**.

> If, for whatever reason, Compose fails to add the changes to the metadata or the mappings, the next time you perform a scan, the changes will be detected again. However, if Compose succeeds to apply the changes but fails to adjust the data warehouse or generate the tasks, the changes will not be detected again. In such a case, you will need to manually adjust the data warehouse and/or generate the tasks after resolving the issue that prevented these operations from being performed automatically.

### Schema evolution using the CLI

You can also use the Compose CLI to perform schema evolution. As opposed to the web console, the CLI does not output a list of changes, which makes it more suited to customers that wish to automate schema evolution through the use of scripts.

To perform schema evolution using the Compose CLI:

1. From the **Start** menu, open the **Compose Command Line** console.

2. Run the following command:
   ```bash
   ComposeCli.exe connect
   ```

3. Run the following command:
   ```bash
   ComposeCli.exe schema_evolution --project project_name [--data_sources mysource1,mysource2,...] --action apply_to_model_and_mappings|apply_to_model_mappings_adjust_storage|apply_to_model_mappings_adjust_storage_generate_tasks|ignore_changes
   ```

   **Where:**
   ```bash
   project_name is the name of your project.
   ```
--data_sources is an optional comma-separated list of data sources to scan for changes. When omitted, all of the project's data sources will be scanned. If there are no changes, “0” will be returned. To avoid errors, make sure the data source names must be exactly as defined in your project.

--action is how (or whether) to apply the changes.

Possible values are:

- apply_to_model_and_mappings - Specify this option if you have other changes you would like to make before adjusting the data warehouse. You may even wish to undo some of the schema evolution changes as Compose does not allow you to choose which changes to apply.
- apply_to_model_mappings_adjust_storage - Specify this option if you have other changes you would like to make before generating the tasks.
- apply_to_model_mappings_adjust_storage_generate_tasks - Specify this option if you want to apply all the changes and generate associated tasks without making any prior changes.
- ignore_changes - Select this option to ignore all changes.

Example:

```
ComposeCli.exe schema_evolution --project project_name --data_sources mysource1,mysource2 --action apply_to_model_mappings_adjust_storage
```

Creating transformations

Compose allows you to transform data using an expression either in Replicate or Compose, according to your needs. The table below provides further information about creating transformations.

<table>
<thead>
<tr>
<th>Where the Transformation is Created</th>
<th>Reasons to Create a Transformation There</th>
<th>When the Transformation is Applied</th>
</tr>
</thead>
</table>
| Replicate                         | • Filtering large amounts of data that is not needed for the Storage Zone (in the present or the future)  
  • Obfuscation due to regulatory reasons or internal policies  
  • Data type conversion (e.g. converting a source data type that is not supported on the Storage Zone platform) | Before the data reaches the landing zone. |
| Metadata                          | • The default location if you are not sure where to put it  
  • General business logic  
  • Needed for several sources or several data marts | Between the Landing Zone and the Storage Zone. |
6  Data Lake projects

<table>
<thead>
<tr>
<th>Where the Transformation is Created</th>
<th>Reasons to Create a Transformation There</th>
<th>When the Transformation is Applied</th>
</tr>
</thead>
<tbody>
<tr>
<td>Storage Zone</td>
<td>• Specific source preparation</td>
<td>Between the Landing Zone and the Storage Zone.</td>
</tr>
<tr>
<td></td>
<td>• Need to preserve the original unfiltered source information in Hadoop</td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Needed for merging several sources</td>
<td></td>
</tr>
</tbody>
</table>

See also Reusable transformations (page 313).

In this section:

- Expression Builder overview (page 308)
- Building Expressions (page 309)
- You test your expression to check that results are as expected. The following figure is an example of an expression that has been evaluated and tested. (page 311)

**Expression Builder overview**

The following section provides an overview of the Expression Builder functionality.

The Expression Builder enables you to create a transformation without needing to type anything manually.

The Expression Builder can be opened in several places, depending on your needs. For more information about where to create a transformation, see Creating transformations (page 307).
The Expression Builder consists of the following panels:

- **Tabs on the left of the Expression Builder**: These tabs contain elements that you can add to an expression. Select elements and add them to the **Build Expression** pane to create an expression. For more information, see *Building Expressions (page 309)*.
  
  The following tabs are available:
  
  - **Parameters** - Only displayed when opening the Expression Builder from within the **Reusable Transformations > Edit Transformation** window. For information on reusable transformations, see *Reusable transformations (page 313)* below.
  
  - **Input Columns/Input Attributes** - Columns/attributes that can be used to build your expression.
  
  - **Transformations** - Contains a list of reusable transformations. The tab is not displayed if no reusable transformations have been defined. For information on reusable transformations, see *Reusable transformations (page 313)* below.
  
  - **Operators** - Operators that can be used to build your expression.
  
  - **Functions** - Functions that can be used to build your expression.

  The Operators and Functions displayed in the Expression Builder use SQL format. As SQL support and implementation is different for each database type and version, the database being used in your Compose project will determine which Operators and Functions will be available.

  Additionally, the list of Operators and Functions displayed in the Expression Builder is not comprehensive. However, you can use any Operators and Functions supported by the database, even if they are not included in the list.

  For an explanation of the available Operators and Functions, refer to the Help for your data lake.

- **Build Expression Pane**: The **Build Expression** pane is where you build your expression. You can add elements, such as columns or operators to the panel as well as type all or part of the expression. For more information, see *Building Expressions (page 309)*.

- **Parse Expression Pane**: This pane displays the parameters for the expression. After you build the expression, click **Parse Parameters** to list the expression parameters. You can then edit the parameters, enter a value for each of the parameters and associate attributes with them. For more information, see *Parsing expressions (page 310)*.

- **Test Expression Pane**: 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 *You test your expression to check that results are as expected. The following figure is an example of an expression that has been evaluated and tested. (page 311)*.

**Building Expressions**

The first step in using the Expression Builder is to build an expression in the **Build Expression** pane.
To build an expression:

1. Hover the mouse cursor over the element that you want to add to your expression (expressions usually start with an Input Column) and click the arrow that appears to its right.
2. Add Operators additional Input Columns and Functions as required.

To add operators to your expression, you can use the Operator tab on the left or the Operator buttons located above the Build Expression pane or any combination of these.

Example:
To create an expression that combines the FirstName name and LastName columns, do the following:

1. Add the FirstName Input Column to the Build Expression pane.
2. In the Operator toolbar above the Build Expression pane, click the concatenate operator.
3. Then add a space between single quote characters and click the concatenate (+) operator again.
4. Add the LastName Input Column to the Build Expression pane.

The expression would look like this:

```
${FirstName}+' '+${LastName}
```

Parsing expressions
When you add operators to the expression, the expression’s parameters are usually added automatically to the Parse Expression pane. However, when you complete your expression or edit it, you may need to parse the expression see all of the parameters.

To parse the expression parameters:

- Click the Parse Expression button below the Build Expression pane.

If the expression is not valid, a red error message will appear at the bottom of the Expression Builder window.

If the expression is valid, the expression parameters and attributes (Input Columns) will be displayed in the in the Parse Expression pane. See the figure Test Expression (page 311).

Editing parameter names
By default, the parameter name is the same as the input column name. However, you can change the parameter name as needed and then associate it with an input column. This is useful, for instance, when you need to shorten attribute names. For example, EstimatedTimeOfArrival can be abbreviated to ETA.
To edit a parameter and associate it with an input column:

1. In the **Parse Expression** pane, edit the parameter name as required.
2. From the **Attribute** drop-down list, select the desired input column.

**Testing expressions**

You test your expression to check that results are as expected. The following figure is an example of an expression that has been evaluated and tested.

*Testing an expression that contains an analytic function will validate the syntax without actually executing the function. Additionally, the test will only be performed on a single record.*

*Compose does not check the data types of columns used in an expression for compatibility. For example, if a column of type integer is used in an expression for a column of type varchar, the expression will not be executed successfully.*

**Test Expression**

1. **Build Expression**

2. **Parse Expression**

3. **Test Expression**

To test an expression:
1. In the Expression Builder window, build an expression as described in Building Expressions (page 309).
2. Click Parse Expression as described in Parsing expressions (page 310).
3. View the parameters that are displayed. If your expression is not valid, an error message is displayed.
4. Optionally edit the parameters name(s) as described in Editing parameter names (page 310).
5. Type values for each parameter and then click Test Expression to see the expression result. For example, using the expression in Test Expression (page 311), type Mike for FirstName and Smith for LastName. The result displayed is Mike Smith.
6. This step is only available for transformations created in the Edit Mappings window. When you create a transformation in the Edit Mappings window, an additional button called Show Data appears to the left of the Test Expression button. You can click this button to see how your expression translates into actual data. For example, clicking the Show Data button for the expression UnitPrice*Quantity will open the following window.
Reusable transformations

In a single Compose project there may be several processes that require similar data transformations. For example a reusable transformation can be defined that concatenates first and last names. This transformation could then be used both in the Customers mapping and in the Employees mapping.
As opposed to stored functions or procedures which are environment dependent, reusable transformations are environment agnostic, meaning that not only can they be used as required within a Compose project, but they can also be used across different environments (using Compose’s export/import function).

Centrally managed transformations increase efficiency by eliminating unnecessary duplication, while at the same time, enabling the seamless propagation of changes to all transformation instances.

Adding a reusable transformation
To define a reusable transformation:

1. From the drop-down menu in the top right of the Storage Zone panel, select Reusable Transformations.
   The Reusable Transformations window opens.
   The window is split into the following panes:
   - Upper pane - Lists the reusable transformations that have been defined.
   - Lower pane - Provides additional information about transformation instances such as where they are in use (e.g. mappings, metadata, etc.) and the expression that was created using the transformation.
   Select a transformation to see the additional information.
2. Click the New Transformation toolbar button.
   The New Transformation window opens.
   1. In the Name field, specify a name for the transformation.
   2. In the Category field, specify a category name. If the category name already exists it will be displayed below the field when you start to type the name. To group the new transformation in the same category, simply select the existing name (unless of course you wish to create a new category with a similar name).
   In the Expression Builder, transformations are grouped according to their category name, making it easier to find the transformation you want to use. Therefore, when specifying a category name, it is recommended to choose a name that reflects the purpose of the transformation. For example, if you create several transformations that concatenate data, it would make sense to group those transformations under a category called "Join".
3. To add a parameter to the transformation, click the New button to the right of the Parameters heading.
   A new row is added to the Parameters list.
4. Specify a name for the parameter, select an appropriate data type, and optionally provide a description.

   If you add multiple parameters, you can change a parameter’s position by selecting the parameter and then using the Up/Down arrows (above the Parameters list) to reposition it.
5. Click the Create Expression button below the Parameters list.
   The Edit Transformation window opens.
6. In the Edit Transformation window, create an expression using the parameters you defined earlier.
For information on creating expressions, see Creating transformations (page 307).

7. Click **OK** to save the transformation.
   The transformation is added to the list in the upper pane.

Once a transformation has been defined, it will be available for selection as needed in the Expression Builder’s **Transformations** tab.

For information on creating expressions, see Creating transformations (page 307).

**Managing reusable transformations**

You can manage reusable transformation as described in the table below.

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delete a transformation</td>
<td>Select the transformation and then click the <strong>Delete</strong> toolbar button. When prompted to confirm the action, click <strong>OK</strong>.</td>
</tr>
<tr>
<td></td>
<td><em>If the transformation is in use, you first need to delete the transformation instances.</em></td>
</tr>
<tr>
<td>Edit a transformation</td>
<td>Double-click the transformation or select the transformation and then click the <strong>Edit</strong> toolbar button. Continue as described in Reusable transformations (page 313).</td>
</tr>
<tr>
<td></td>
<td><em>Any changes you make to a transformation will be propagated to all instances of that transformation.</em></td>
</tr>
<tr>
<td>Edit a parameter</td>
<td>Open the <strong>Edit Transformation</strong> window as described in Reusable transformations (page 313). Then, select the parameter you want to delete and click the <strong>Delete</strong> button above the Parameters list.</td>
</tr>
</tbody>
</table>

**6.6 Creating and Managing Storage Zone Tasks**

Once the Metadata has been prepared, the next step in the Compose workflow is to create the Storage Zone tables (optional), generate the task statements and run the Storage Zone task. Tasks can either be run manually or scheduled to run periodically or in the future.

In this section:

- Defining and running data storage tasks (page 316)
- Managing task definitions (page 319)
- Viewing and exporting task statements (page 329)
- Modifying task settings (page 329)
Defining and running data storage tasks

This section describes how to create the Storage Zone tables, generate the task statements and run a Data Storage task.

It contains the following topics:

- Creating the Storage Zone (page 316)
- Generating the task statements (page 316)
- Controlling data storage tasks (page 317)

Creating the Storage Zone

If table(s) were reloaded in Replicate, the table(s) will also be (automatically) reloaded in Compose the next time the task runs. During reload, Live Views should not be read.

To create the Storage Zone tables:

1. Click the Create button in the bottom right of the Storage Zone panel.
   The Creating Storage Zone window opens.
   A progress bar indicates the current progress. For each stage of the Storage Zone generation process, a corresponding message appears in the Messages list.
2. When the process completes, click Close.

See also: Supported characters (page 370).

Generating the task statements

After the Storage Zone tables have been created, you then need to generate the task statements that will be used in the Storage Zone task. The task statements include the Mappings between the Landing Zone tables and the Storage Zone tables. If you need to make changes to the Mappings, continue from Managing task definitions (page 319).

- Changing a Primary Key in the source record will cause a new record to be inserted in the storage table.
- Regenerating the task statements after performing a non-supported change in the metadata will appear to succeed without errors or warnings, but the task will fail if run later.
- Defining a single task that ingests data from several Landing Zones is not supported. As a workaround, you can create a separate task for each Landing Zone.

To generate the Storage Zone task statements:

1. Click the Data Storage Tasks button in the bottom left of the Storage Zone panel.
   The Manage Data Storage Tasks window opens.
2. If there are multiple tasks, in the left pane, select the desired task.
3. Click the **Generate** toolbar button, and select one of the following options:

- **With validation** - This is the default option for generating storage tasks.

  You can also generate storage tasks with validation (default option) by clicking the **Generate** toolbar button.

- **Without Validation** - Select this option if you are sure that the storage tables are adjusted properly and the mapping is valid. The generation of storage tasks is much faster. Note that you could have errors later when running the task if something is not valid.

  The **Generating Statements for Task: Name** progress window opens. When the "Generate task finished successfully" message is displayed, close the window.

  Only mappings associated with the task in the **Manage Tasks** window will be generated.

**Controlling data storage tasks**

Once the Storage Zone tables have been created and the task statements have been generated, you can then proceed to run the Storage Zone task. The Storage Zone task extracts data from the Landing Zone tables and loads it into the Storage Zone tables.

**Limitations and considerations**

- When task are reloaded in Replicate November 2020, Compose will also reload the task (i.e. perform a Full Load) and only then apply the changes. Depending on the number of tables involved, this may take some time as two reloads will be performed (one in Replicate and the other in Compose).
- A storage directory may be used exclusively by only one Compose project.
- Data storage tasks are optimized to run on relatively large batches of data. It is recommended to specify a partition length in excess of one hour. Although specifying a partition length of 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).
- Change Processing creates a new file on every write. This may cause many files to amass and degrade performance. Therefore, it is recommended to monitor the storage directory and periodically consolidate small files into larger ones and move/delete files that are no longer required.
- Storage directories and subdirectories are managed by Compose; you should not delete files or write to them unless approved by Qlik Support or explicitly recommended in this guide.
- When using a Hive-based compute platform, for optimal performance, it is recommended to allocate a dedicated queue to Compose tasks only.
- When using a Hive-based compute platform, in order to see the delta of data changes in the storage tables, you need to define the following commands so that Hive can read the subdirectories:

```
set hive.supports.subdirectories=true;
set hive.input.dir.recursive=true;
```
Running a task

Storage Zone tasks can be run manually, scheduled to run periodically or run as part of a workflow. The section below describes how to run a Storage Zone task manually. For information on scheduling Storage Zone tasks or including them in a workflow, see Controlling and monitoring tasks and workflows (page 331).

If there is a Replicate source table with data, that:

- Was not originally selected in the Replicate Full Load and Apply Changes task (i.e. was added later).

-OR-

- Was selected in a Replicate Full Load and Apply Changes task, but was not selected in the mappings of the Compose Full Load and Change Processing data storage tasks, and the tasks have already been run.

In any of the above scenarios, in order to get the data that was added later, you need to:

1. Duplicate the Compose Full Load and Change Processing tasks associated with that table.
2. Run the duplicated Full Load task.
3. Run the duplicated Change Processing task.

Note that after running these tasks, duplicate records may exist in the Storage Zone, but they will be removed when reading from the Storage Zone views.

To run a Storage Zone task:

1. Click the Manage button in the bottom left of the Storage Zone panel.
   The Manage Storage Tasks window opens.
2. If multiple tasks have been defined, in the left pane, select the task that you want to run.
   Click the Run toolbar button. The window switches to Monitor view and the following status bars are displayed:
   - Completed - Shows the tables that have already been loaded into Hive
   - Loading - Shows the tables currently being loaded into Hive
   - Queued - Shows the tables waiting to be loaded into Hive
   - Error - Shows the tables that could not be loaded into Hive due to error. Click the Show Details link below the bar to see more information about the statement(s) that resulted in the error.
   - Canceled - The number of canceled tables (tables that were not processed due to the task being aborted) does not appear as a separate status bar. To view the number of canceled tables, click the Select All link above the status bars.
3. To see more information about tables in a particular status, click the relevant status bar. A list of tables in the selected status will be shown.
Managing task definitions

Task definitions contain the mappings between the columns in the Landing Zone tables and the columns in the Storage Zone tables (and any transformations applied to those mappings). The same mappings can be used by several tasks. You can create new tasks, duplicate tasks and edit existing tasks as required.

The following options are available:

- Adding and duplicating tasks (page 319)
- Column mappings (page 320)

  - For each Compose task, all of the mapping tables should be populated by data from one Replicate task.
  - You must regenerate the task statements and then run a Storage Zone task whenever the mappings are modified. Populating the Storage Zone can either be done manually as described in Controlling data storage tasks (page 317) or automatically as described in Scheduling tasks (page 334). If you have already run the data mart tasks, then you also need to regenerate the data mart ETLs and run the tasks again as described in Managing task definitions (page 319).

- Using lookup tables (page 324)
- Dropping and recreating tables (page 326)

Adding and duplicating tasks

As the default task definitions are generated automatically (by discovering the Landing Zone tables), there is usually no reason to manually create or duplicate an task. One possible reason to duplicate an task is if your Metadata contains different types of tables and you want to manage them in separate tasks. An exception to this is if you import your model from ERwin without first defining global mappings. In such a case, you will need to manually add the Task and create the mappings.

For more information on global mappings, see Managing global mappings (page 147).

The following task types are available:

- **Full Load** - Loads the selected tables from the Landing Zone into the Storage Zone.
- **Change Processing** - Updates the Storage Zone tables with the Landing Zone changes.
• **Full Load and Change Processing** - Loads the selected tables into the Storage Zone and then updates them with the Landing Zone changes.

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add a new Task</td>
<td>1. Click the <strong>Manage</strong> button at the bottom left of the <strong>Storage Zone</strong> panel. The <strong>Manage Tasks</strong> window opens.</td>
</tr>
<tr>
<td></td>
<td>2. Click the <strong>New Task</strong> toolbar button. The <strong>Add New Task</strong> window opens.</td>
</tr>
<tr>
<td></td>
<td>3. Specify a name for the task.</td>
</tr>
<tr>
<td></td>
<td>4. Optionally, specify a description.</td>
</tr>
<tr>
<td></td>
<td>5. Choose <strong>Full Load</strong> and/or <strong>Change Processing</strong> as the task type.</td>
</tr>
<tr>
<td></td>
<td>6. Click <strong>OK</strong>.</td>
</tr>
<tr>
<td></td>
<td>7. Select the task name in the left pane and continue from <strong>Column mappings</strong> <em>(page 320)</em>.</td>
</tr>
<tr>
<td>Duplicate a Task</td>
<td>1. Click the <strong>Manage</strong> button at the bottom left of the <strong>Storage Zone</strong> panel. The <strong>Manage Tasks</strong> window opens.</td>
</tr>
<tr>
<td></td>
<td>2. Select the task you want to duplicate and then click the <strong>Duplicate</strong> toolbar button. The <strong>Duplicate</strong> window opens.</td>
</tr>
<tr>
<td></td>
<td>3. Specify a <strong>Name</strong> for the new Task.</td>
</tr>
<tr>
<td></td>
<td>4. Select a Landing Zone.</td>
</tr>
<tr>
<td></td>
<td>5. Optionally change the default <strong>Schema</strong>.</td>
</tr>
<tr>
<td></td>
<td>6. Select the <strong>Task type</strong> according to your Replicate task type.</td>
</tr>
<tr>
<td></td>
<td>7. Click <strong>OK</strong>.</td>
</tr>
<tr>
<td></td>
<td>8. Select the task name in the left pane and continue from <strong>Column mappings</strong> <em>(page 320)</em>.</td>
</tr>
</tbody>
</table>

**Column mappings**

*For improved metadata performance during discovery and mapping, Compose caches the metadata of the Landing Zones after reading them. However, synchronization issues may arise if the Landing Zone is modified outside of Compose. In such cases, you should click **Clear Landing Cache** in the **Mappings** tab of the **Storage Zone** panel in order to refresh the cache on the next reading of the metadata.*

*For details on recreating the Storage Zone cache, see **Clearing the metadata cache** *(page 328).*

The mappings show the current mapping between the Landing Zone tables and the Storage Zone tables. By default, the columns names and data in the Landing Zone tables and the Storage Zone tables will be identical. However, you can manually change the mappings according to your needs, either by simply mapping a Landing Zone column to a different Storage Zone column and/or by using an expression.
Limitations and considerations

- Creating multiple mappings for a single table is not supported.
- Mapping from Views is not supported.

Editing column mappings

To edit column mappings:

1. Click the Manage button in the Storage Zone panel.
   The Mappings tab is displayed. Each of the Storage Zone tables has a corresponding mapping name.
2. In the Mappings column, click the mapping that you want to edit.
   The Edit Mapping: Name window opens.
3. Edit the mapping as described in the table below.
<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
<tbody>
<tr>
<td>Map a column in a Landing Zone table to a column in a Storage Zone table</td>
<td>The mapping procedure differs depending on whether you are in Standard View or Compact View. For information on changing the view, see Change the view (page 322).</td>
</tr>
</tbody>
</table>

**In Standard View:**

1. Hover the mouse cursor over the Landing Zone column name as shown in the image below. A gray dot appears to the right of the column name.

   ![Image](image1.png)

2. Drag the mouse cursor from the gray dot to the desired column in the Storage Zone table.

   ![Image](image2.png)

3. When the dotted line turns green (as shown below), release your mouse button. The mapping operation is completed. Note that if the dotted line turns red (instead of a green), you will not be able to map the Landing Zone column with the desired Storage Zone column. A red dotted line indicates that the Landing Zone and Storage Zone column data types are incompatible with each other.

**In Compact View:**

1. Switch to Compact View as described in Change the view.
2. Drag the Landing Zone column to the cell located to the left of the target Storage Zone column.

**Auto-generate mapping**

Click the **Auto-Map** toolbar button.

**Remove all mappings**

Click the **Reset** toolbar button.

**Change the view**

Changing to a more compact view is recommended for Landing Zone tables that have numerous columns. In compact view, the table columns are organized in rows (instead of a single list), making it easier to locate Landing Zone columns and map them to the desired Storage Zone columns. **To change the view:**

Click the **Change View** toolbar button. For information on creating mappings in Compact view, see Map a column in a Landing Zone table to a column in a Data Lake table.
To | Do This
---|---
Select a different source database | Select a database from the **Landing Zone Database** drop-down list on the left of the window.
Select a different source schema | Select a schema from the **Schema** drop-down list on the left of the window.
Select a different table | Select a table from the **Table** drop-down list on the left of the window.
Create a column-level transformation | 1. Hover the mouse cursor over the Storage Zone Column for which you want to create a transformation and then click the **fx** button that appears to its right. The Expression Builder opens.
2. Continue from *Creating transformations (page 307).*

Adding, renaming and deleting mappings
You can add, rename and delete mappings as required. For example, if you want one of the Storage Zone tables to contain columns from several tables in the Landing Zone, then you need to add a new mapping for each of the Landing Zone tables.

To add, delete, and rename mappings:

1. Click the **Manage** button in the **Storage Zone** panel. The **Manage taskTasks** window opens.
2. In the left pane, select the desired taskTask.
3. Select the **Mappings** tab.
4. Add or delete mappings as described in the following table.

To | Do This
---|---
Add a new mapping | 1. In the **Data Lake Tables** column, select the table that you want to map.
2. Click the **NewMapping** button above the **Delivery Tables** column. The **New Mapping** window opens.
3. Optionally change the default mapping name.
4. From the **Entity** drop-down list, select the entity in the Storage Zone to which you want to map.
5. Click **OK** to save the mapping.
6. Enable the mapping.
Delete a mapping | 1. In the **Mappings** column, hover the mouse cursor over the mapping you want to delete.
2. Click the **Delete(x)** button that appears to its right.
3. Click **OK** when prompted to confirm the deletion.
Using lookup tables

Lookup tables are useful for replacing source data with the actual data that you want to appear in the Storage Zone. For example, a lookup table could be used to replace a zip code with a full address or, conversely, to replace a full address with a zip code.

**Lookup on a column which is mapped to the Compose “From__Date” column is not supported.**

Linking lookup tables

To link a lookup table column to a Storage Zone table column:

1. Click the link to the desired task in the Storage Zone panel. The Manage Tasks window opens.
2. In the Mappings column, click the mapping for the Storage Zone table containing the result column (with the data that you want to replace). The Edit Mapping - Name window opens.
3. Hover the mouse cursor over the relevant Storage Zone column and then click the Lookup button that appears to the right of the column name. The Select Lookup Table window opens.
   1. From the Database drop-down list, select the database containing the lookup table.
   
   **The database must reside in your Landing Zone.**

   2. From the Schema drop-down list, select the schema containing your source lookup tables.

   3. Select either Table or View according to the lookup table type.

   4. From the Table drop-down list, select the lookup table. The right side of the Select Lookup Table window displays the lookup table columns and their data types. To view the data in the lookup table, click the Show Lookup Data button.

   5. After you have selected the lookup table, click OK.

The Lookup Transformations - Table Name.Column Name window opens. The window is divided into the following panes:

**Upper pane:** The upper part of the right pane (Condition) displays the condition expression, which stipulates the condition(s) for performing the lookup.

**Lower pane:** The lower part of the right pane (Result Column) displays the column result expression, which stipulates what data to replace in the target column.

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
</table>
| Rename a mapping    | 1. In the Mappings column, hover the mouse cursor over the mapping you want to rename.  
2. Click the Rename (A) button that appears to its right. The Rename window opens.  
3. Specify a new name for the mapping and then click OK. |
4. To change the lookup table, click the **Change Lookup Table** button above the lookup table columns and then perform steps a. to d. above.

5. To view the lookup table or landing table data, click the **Show Lookup Data** or **Show Landing Data** buttons respectively.

6. To specify the condition(s) for performing the lookup, click the **Create Expression** button (which changes to **Edit Expression** after an expression has been created) above the **Condition** expression.
   
   The **Condition Expression - Column Name** window opens.

7. Create an expression using the landing and lookup table columns on the left.

   For an example, see **Using lookup tables** (page 324).

   For information on creating expressions, see **Creating transformations** (page 307).

8. To specify what data to replace or add if the lookup conditions are met, click the **Create Expression** button (which changes to **Edit Expression** after an expression has been created) above the **Result Column** expression.

   The **Result Expression - Column Name** window opens.

9. Create an expression using the landing and lookup table columns on the left.

   For an example, see **Using lookup tables** (page 324).

   For information on creating expressions, see **Creating transformations** (page 307).

10. To preview the results, click the **Preview Results** button.

11. Click **OK** to save your settings and close the **Lookup Transformations - Table Name Column Name** window.

### Lookup example

The following example shows how a lookup table is used to concatenate a Dutch translation of the category name (located in the lookup table) to the original category name located in the landing table.

The lookup could be defined using the following expressions:

1. **Condition expression**:

   \[
   \{\text{Lookup.CategoryID}\} = \{\text{Landing.CategoryID}\}
   \]

   **Meaning**: Perform the lookup only if the Category ID in the landing table and the lookup table are the same.

2. **Result column expression**:

   \[
   \{\text{Lookup.CategoryName}\} + \ ' \ is ' + \{\text{Landing.CategoryName}\}
   \]

   **Meaning**: Add the data in the **CategoryName** column in the lookup table to the data in the **CategoryName** column in the landing table (separated by the word "is").

Assuming the result column name is "Split Name", clicking the **Preview Results** button would display the following table:

<table>
<thead>
<tr>
<th>Split Name</th>
<th>Category Name (Lookup)</th>
<th>Category Name (Landing)</th>
<th>Category ID (Lookup)</th>
<th>Category ID (Landing)</th>
</tr>
</thead>
<tbody>
<tr>
<td>dranken is Beverages</td>
<td>dranken</td>
<td>Beverages</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Specerijen is Condiments</td>
<td>Specerijen</td>
<td>Condiments</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>Split Name</td>
<td>Category Name (Lookup)</td>
<td>Category Name (Landing)</td>
<td>Category ID (Lookup)</td>
<td>Category ID (Landing)</td>
</tr>
<tr>
<td>------------------------------------</td>
<td>------------------------</td>
<td>-------------------------</td>
<td>----------------------</td>
<td>----------------------</td>
</tr>
<tr>
<td>Gebak is Confectionary</td>
<td>Gebak</td>
<td>Confectionary</td>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td>Zuivelproducten is Dairy Products</td>
<td>Zuivelproducten</td>
<td>Dairy Products</td>
<td>4</td>
<td>4</td>
</tr>
<tr>
<td>Grains/Granen is Grains/Cereal</td>
<td>Grains/Granen</td>
<td>Grains/Cereal</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td>Vlees/Gevolgete is Meat/Poultry</td>
<td>Vlees/Gevolgete</td>
<td>Meat/Poultry</td>
<td>6</td>
<td>6</td>
</tr>
</tbody>
</table>

### Dropping and recreating tables

Compose enable you to drop and recreate the Storage Zone tables as required.

When changing certain project settings (e.g. table prefixes) drop and create is required. If you change the Metadata after the Storage Zone tables and/or files were already created and loaded with data, you should adjust the Storage Zone to reflect the modified Metadata (as described in *Validating the metadata and storage (page 298)*). Some changes however cannot be resolved by adjusting the Storage Zone. In such cases, you can either revert the Metadata to its pre-modified state or drop and (optionally) recreate the Storage Zone tables.

Note that dropping and recreating tables will delete all of the data in the tables and should only be performed in lieu of a better option.

- **In some scenarios, you need to edit the CREATE table statements before they are run. This can be done using the Generate DDL scripts but do not run them option in Project settings (page 38).** For example, if you want to override the default sorting of your Storage Zone tables or add specific formatting annotations, you will need to edit the script to accomplish this.

- **The Change Processing context (i.e. the point in time when changes were last captured) is deleted when dropping all tables but preserved when dropping selected tables. Therefore after deleting selected tables, in order for Compose to continue processing changes from when the tables were dropped, you need to perform the following additional steps:**
  1. Duplicate the Full Load and Change Processing tasks.
  2. Delete the old tasks (i.e. the tasks that were duplicated) making sure to select the Delete mappings not used by other tasks option.
  3. Run the Full Load and Change Processing tasks again. This may result in duplicates in the Storage Zone tables, but the duplicates are excluded from the Storage Zone Views and will not appear in any provisioned data.

To drop and recreate all storage tables:
1. In the Storage Zone panel, select the **Drop and Recreate|Tables** item from the menu in the top right corner.

2. The **Drop and Recreate Tables** window opens.

3. Select **Recreate** to drop and recreate the storage tables or **Drop** to drop them only.

4. Click **OK** to perform the drop and/or recreate operation.

   The tables will be dropped and/or recreated, unless the **Generate DDL scripts but do not run them** option is enabled.

**Dropping and recreating views**

Generally speaking, you should not need to recreate the views very often. Usually, recreating the views is only required after upgrading to a newer Compose version or patch release that contains updates to the views. Although unlikely, you may also need to recreate the views if they were accidentally deleted. If there is a need to recreate the views after upgrading, it will be clearly stated in the release notes. The Views can be recreated using the Compose web console or using the Compose CLI.

If Compose detects a mismatch between the Logical Metadata (defined via the **Metadata** panel) and the Storage Zone metadata, the view recreation operation will fail and you will need to validate and adjust the storage before retrying the operation.

**Recreating the Views with the web console**

To recreate the Views using the web console, simply select **Recreate Views** from the menu in the top-right of the **STORAGE ZONE** panel. You will be prompted to confirm the operation as it might take some time, during which the Views data might not be accessible.

**Recreating the Views with the CLI**

To recreate the Views using the CLI:

1. From the **Start** menu, open the **Compose Command Line** console.

2. Run the following command to connect to the server:

   ```
   ComposeCli.exe connect
   ```

3. When the command completes successfully, run the following command:

   ```
   ComposeCli.exe recreate_views --project project_name
   ```

   Where:

   - `--project` is the name of the project for which you need the views to be recreated.

   **Example:**

   ```
   ComposeCli.exe recreate_views --project myproject
   ```

**See also:** *Validating the metadata and storage (page 298)*
Clearing the metadata cache

To improve performance when reading from the Landing Zone or from the Storage Zone, Compose caches both the Landing Zone metadata and the Storage Zone metadata. However, synchronization issues may sometimes occur if the structure of the Landing Zone or the Storage Zone metadata is altered outside of the Compose project.

If you aware of external changes to the metadata or if you notice any data synchronization anomalies, Compose enables you to clear the metadata cache, either using the web console or using the CLI.

Clearing the landing zone metadata cache

To clear the landing zone metadata cache and refresh the mappings on the next reading of the metadata:

1. Click the Manage button at the bottom left of the Storage Zone panel.
2. Click the Clear Landing Cache button in the Manage Storage Tasks window.

See also the section describing how to clear the cache before discover.

Clearing the storage zone metadata cache

To clear the storage zone metadata cache:

1. In the Storage Zone panel, select the Clear Metadata Cache item from the menu in the top right corner.
2. Click Yes to clear the storage zone metadata.
3. When the storage zone metadata cache has been successfully cleared, click Close.

Clearing the metadata cache using the CLI

To clear the metadata cache using the CLI:

1. From the Start menu, open the Compose Command Line console.
2. Run the following command:
   
   ```powershell
   ComposeCli.exe connect
   ```
3. Run the following command:
   
   ```powershell
   ComposeCli.exe clear_cache --project project_name [--type landing|storage] [--landing_zone source_name]
   ```

   Where:
   - `--project` is the name of your project.
   - `--type` is where to clear the metadata cache. Possible values are landing or storage. If `--type landing` and you want to clear a specific landing zone, you must set the `--landing_zone` parameter as well. To clear the metadata cache in all landing zones, specify `--type landing` and omit the `--landing_zone` parameter.
   - `--landing_zone` is the name of the landing zone when `--type landing`. 
Example:

```
ComposeCli.exe clear_cache --project myproject --type landing --landing_zone mysourc1
```

Viewing and exporting task statements

You can view the task statements that were run during the Storage Zone task. You can also export the task statements to a TSV file for reviewing and sharing.

To view the task statements:

Click the Task Statements toolbar button.

The Task Statements - `<Source_Landing_Zone_Name>` window opens in List View. The Description column provides a description of each operation that was performed on the target tables. Each operation has a different process number (displayed in the Process Number column). For additional details about an operation, double-click the operation.

-OR-

Click the Item View button and navigate through the statements using the navigation buttons at the bottom of the Task Statements - `<Source_Landing_Zone_Name>` window.

To jump to a specific statement, type the statement's number in the Go To field at the bottom of the window and then press [Enter].

To export the task statements to a TSV file:

In List View, click the Export to TSV File button located to the left of the search field.

A file named "<name>_Task_Instructions.tsv" will be saved to your default "Downloads" location or you will be prompted to save it (according to your browser settings).

To hide non-SQL steps from the display, select the Filter non-SQL steps check box

Modifying task settings

For each task, you can modify the settings according to your needs.

To modify task settings:

1. In the Manage Storage Tasks window, select a task in the left pane and then click Settings. The Setting - `<Task_Name>` window opens.
2. In the General tab, you can change the logging granularity. In the Log level drop-down list, the following options are available:
   - INFO (default) - Logs informational messages that highlight the progress of the task at a coarse-grained level.
   - DEBUG - Logs fine-grained informational events that can be used to debug the task.
• **TRACE** - Logs finer-grained informational events than the **DEBUG** level. Note that the log levels **DEBUG** and **TRACE** impact performance. You should only select them for troubleshooting if advised by Qlik Support.

3. In the **Advanced** tab, the following settings are available:

   • **Sequential Processing**: Select this option if you want all the Storage Zone processes to run sequentially, even if they can be run in parallel. This may be useful for debugging or profiling, but it may also affect performance.

   • **Maximum number of database connections**: Enter the maximum number of database connections that Compose is allowed to open for the task. The default number is 10.

   • **JVM memory**: Edit the memory for the java virtual machine (JVM) if you experience performance issues. **Xms** is the minimum memory; **Xmx** is the maximum memory. The JVM starts running with the Xms value and can use up to the Xmx value.

   • **Position in default workflow**: Select where you want the Storage Zone tasks to appear in the default workflow. For more information on workflows, see *Workflows (page 337)*.

4. To save your changes, click **OK**.

### 6.7 Creating and managing command tasks

Command tasks enable you to incorporate custom processes into your Compose workflow. This is especially useful if you need to leverage external tools to transfer files, validate data, and so on. A Command task can run any script or executable supported by the operating system including batch files, Python scripts, PowerShell scripts, executables and so on.

In this section:

- **Defining Command tasks (page 330)**
- **Managing Command tasks (page 331)**
- **Controlling and monitoring Command tasks (page 331)**

### Defining Command tasks

This section explains how to define a command task. You can define as many command tasks as you need and execute them at different stages of a Compose workflow.

> For security reasons, before you define a command task, make sure that the executable or script file that you want to run resides in the following directory on the Compose server machine:

```
PRODUCT_DIR\data\projects\YOUR_PROJECT\scripts
```

To define a command task:

1. From the project drop-down menu, select **Manage Command Tasks**. The **Manage Command Tasks** window opens.
2. Provide a name for the task.
3. Optionally, enter a description.
4. In the **Script/Executable File** field, specify the name of the files that you want to run.

5. In the **Parameters** field, specify any parameters required by the command. Parameters should be separated by a space.

6. The user context is the user account under which the Task will run. To change the current user context, provide the **User, Password** and **Domain** of the account under which you want the Task to run.

7. Click **Save** to save your changes or **Discard** to discard any unsaved changes. The task will be added to the list of tasks in the left of the window.

Managing Command tasks

The table below describes the task management options.

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
<tbody>
<tr>
<td>Edit a task</td>
<td>Select the task in the tasks list in the left of the <strong>Manage Command Tasks</strong> window and edit it as described in <em>Defining Command tasks</em> (page 330).</td>
</tr>
<tr>
<td>Delete a task</td>
<td>Select the task in the tasks list in the left of the <strong>Manage Command Tasks</strong> window and then click the <strong>Delete</strong> toolbar button. When prompted to confirm the deletion, click <strong>OK</strong>.</td>
</tr>
<tr>
<td>Search for a task</td>
<td>Enter part of the task name in the search box above the task list. The list of tasks will be filtered to show only tasks that include the search term in their name.</td>
</tr>
</tbody>
</table>

Controlling and monitoring Command tasks

Command Tasks can be run from the **Manage Command Tasks** window or from the main Compose Monitor view. Although they can be run individually, command tasks are usually run as part of a workflow.

For information on defining workflows, controlling and monitoring tasks, and controlling and monitoring workflows, see *Controlling and monitoring tasks and workflows* (page 331).

To run a command task from the Manage Command Tasks window:

1. Open the **Manage Command Tasks** window and select the task you want to run.
2. Click the **Run** toolbar button.
3. The **Manage Command Tasks** window switches to **Monitor** view.

   In Monitor view the following information is available:
   
   - The task ID
   - The current status
   - When the task started and ended
   - The overall task progress

6.8 Controlling and monitoring tasks and workflows

The Compose monitor shows the current status of all your tasks and enables you to drill-down for additional information about each task. Task instances can be run immediately or scheduled to run in the future (either once or at set intervals).
In this section:

- **Viewing information in the monitor (page 332)**
- **Running and controlling tasks (page 333)**
- **Notifications (page 335)**
- **Workflows (page 337)**
- **Monitoring and controlling Replicate tasks (page 341)**

### Viewing information in the monitor

As well as providing a high-level summary of all your tasks, the monitor also lets you view more detailed information about specific tasks.

To switch to monitor view:

1. Open a Compose project and click the **Monitor** icon in the top right of the console.
   A list of tasks is displayed for the current project. The left pane of the monitor allows you to filter the task list by status as well as indicating the current number of running, failed and completed tasks.

For each task, the monitor displays the following information:

- **Status** - Running, Completed, Failed or Aborted.
- **Task** - The task name.

**Type** - The following task types are available:

- **Storage Full Load** - Moves the data in its entirety from the Landing Zone to the Storage Zone.
- **Storage Change Processing** - Moves changes to the data from the Landing Zone to the Storage Zone.
- **Workflow** - Executes several tasks in succession. See also *Adding and designing workflows (page 338)*.
• Command - For information about Command Tasks, see Creating and managing command tasks (page 330)
• Replicate - The Qlik Replicate task that moves the data from the source database to the Landing Zone.
• Started and Ended - The date and time the task started and completed (according to the server time). If the task is running, the Ended column will display the current progress. In the case of a Replicate task performing Change Processing, Running CDC will be displayed.
• Next Instance - The next time the task is due to run (if the task is scheduled).
• Elapsed Time - The time it took for the task to complete or - if the task is still running - how long the task has been running.
• Updated Tables - The number of tables updated in the Storage Zone.
• Scheduled - Whether the task has been scheduled. "N/A" indicates that the task has never been scheduled whereas a check box indicates that the task has been scheduled. Clear the check box to disable the scheduling.

2. To view additional information about a task, select the task. The information is displayed in the following tabs in the lower pane of the monitor:
   • Details - The Details tab shows the following status bars:
     • Completed - Shows the tables that have already been loaded into Hive.
     • Loading - Shows the tables currently being loaded into Hive.
     • Queued - Shows the tables waiting to be loaded into Hive.
     • Error - Shows the tables that could not be loaded into Hive due to error. Click the Show Details link below the bar to see more information about the statement(s) that resulted in the error.
   To see more information about tables in a particular status, click the status bar. A list of tables in the selected status will be shown.
   You can also click the Task Commands button for more information about the operations performed during the task.
   • Progress Status - The Progress Status tab shows the task's current progress as well as the sub-status (Running, Failed, etc.) of operations within the task. To see details about a specific operations, click the number to the right of the operation status. For example, to view more information about an operation with an error status, click the number to the right of the Failed bar.
   • History - The History tab provides a list of previous task instances. To view a task instance’s log file, select the task and click the View Log button.
   To view more details about a task instance, either double-click the instance or select the instance and then click the View Instance Details button. The Details tab is shown.

3. To run a task immediately, select the task and then click the Run toolbar button.
4. To view and manage a task’s settings, select the task and then click the Open toolbar button. For more information about the settings, see the relevant topic in this guide.

Running and controlling tasks

You can run and stop tasks/workflow manually or you can schedule them using the scheduling options described in Scheduling tasks (page 334).
Running and aborting tasks manually
You can run tasks manually and abort them if required.

To run a task manually:
- Select the task and click the Run toolbar button.

To abort a task:
- Select the task and click the Abort toolbar button.
  The task process is aborted. Note that aborting a task may leave the Storage Zone or data mart tables in an inconsistent state. Consistency will be restored the next time the task is run.

Scheduling tasks
Scheduling tasks is a convenient way of continually updating the Storage Zone and associated data mart(s). For instance, you could schedule a data warehouse task to run at 4:00 pm and then schedule a data mart task to run at 5:00 pm.

Note that as Compose currently does not provide a task-chaining option (i.e. run another task as soon as the current task completes), it may be better to schedule tasks using an external tool that supports this capability.

You can also use the command line interface (CLI) to run a task. For details, see Running tasks using the CLI (page 335).

To schedule a task:

1. Click the Schedule toolbar button.
2. In the <Name> Scheduler window, choose one of the following options from the Run Job dropdown list.
   - Once - to run the job once on a specific date and time.
   - Every - to run the job at set intervals.
   - Daily - to run the job every day at a specific time.
   - Weekly - to run the job on selected days at a specific time
   - Monthly - to run the job on the nth of every month at a specific time
   - Advanced - to use a Cron expression. For a description of allowed cron formats together with usage examples, see Cron format and examples (page 367).
3. Set the scheduling parameters according to the selected scheduling option.
4. Click OK to save your settings.
   The date and time the next instance is scheduled to run will appear in the Next Instance column.
5. To disable a scheduled job, select the task and click the Edit Scheduling toolbar button. Then, select the Disable check box in the <Name> Scheduler window.
6. To cancel a scheduled job for a task, select the task and click the Edit Scheduling toolbar button. Then, in the <Name> Scheduler window, click Delete.
Running tasks using the CLI

You can also run tasks using the CLI. This is especially useful if you wish to run Compose tasks from external schedulers such as HP OpenView or Control-M. Before you can run a task, you must first run the Connect command as described in Connecting to Qlik Compose server (page 76).

As Compose CLI requires Administrator permission, make sure to select "Run as administrator" when opening the command prompt.

The run_task command populates the Storage Zone with data. The task can also be run using the Run toolbar button located in Monitor view as well as in the Manage Task window.

When this command succeeds, it returns 0.

Syntax

```
ComposeCli.exe run_task --project [project-name] --type [storage|workflow] --task [task-name] --wait [timeout-in-sec]
```

where:

- **project** is the name of your Compose project
- **type** is the type of task you want to run.
  - *storage* - Data Storage task
  - *workflow* - Workflow task
- **task** is the name of the task you want to run.
- **wait** is the wait time specified in seconds.

The command line can run in sync or async mode. A value of 0 (seconds) indicates sync mode. This means that as soon as the task finishes, the command line returns to prompt. The default mode is async, with a value of -1. This is also applied if you leave this parameter empty. Other negative values are not permitted.

Note that if **wait** is excluded from the command, the task may appear to complete successfully even if it encountered an error.

Example

```
ComposeCli.exe run_task --project nw --task hive_analytics --type storage
```

Compose Control Program started...
Instance Number: 9
Compose Control Program completed successfully.

Notifications

You can select events, on the occurrence of which, a notification will be sent to the specified recipients.

Notifications will not be sent unless the mail server settings are correctly defined.
Setting notifications

To set a notification rule:

1. Switch to Monitor view.
2. Click the Notifications toolbar button.
   The Notification Rules window opens.
3. Click the New toolbar button.
   The New Notification wizard opens.
4. In the Events screen:
   • Specify a name for the notification
   • Choose for which type of events you want the notification to be sent, both at the task level and at the workflow level.
5. Click Next. In the Recipients screen:
   • Select Windows Event to send the notification to Windows Event Log and/or Recipients to send the notification to a list of email recipients. See also: Notifications (page 335).
   • If you selected Recipients, enter the recipient email addresses in the To, Cc (optional) and Bcc (optional) boxes. Multiple addresses must be separated by a semi-colon.
6. Click Next. In the Message screen, optionally, edit the default notification message. You can add variables to the message by selecting the variable on the right and then clicking the arrow to the left of the variables list.

The following variables are available:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>${PROJECT}</td>
<td>The name of the Compose project in which the event occurred.</td>
</tr>
<tr>
<td>${TASK_NAME}</td>
<td>The name of the task in which the event occurred.</td>
</tr>
<tr>
<td>${INSERTED}</td>
<td>The number of rows inserted in the Storage Zone.</td>
</tr>
<tr>
<td>${UPDATED}</td>
<td>The number of rows updated in the Storage Zone.</td>
</tr>
<tr>
<td>${DELETED}</td>
<td>The number of rows deleted from the Storage Zone.</td>
</tr>
<tr>
<td>${ERROR_CODE}</td>
<td>The error code if an error was encountered during the task.</td>
</tr>
<tr>
<td>${ERRORDETAILS}</td>
<td>The error message if an error was encountered during the task.</td>
</tr>
<tr>
<td>${EVENT_TYPE}</td>
<td>The event type (Started, Error or Completed).</td>
</tr>
<tr>
<td>${EVENTTYPEDESCRIPTION}</td>
<td>The event type (Started, Error or Completed).</td>
</tr>
<tr>
<td>${EVENT_TIME}</td>
<td>The date and time the event occurred.</td>
</tr>
<tr>
<td>${LINK}</td>
<td>A link to the relevant Compose project.</td>
</tr>
</tbody>
</table>
7. Click **Next**. In the **Apply to** screen, select whether to apply the rule to all tasks of to selected tasks. If you chose **Selected Tasks**, select which tasks to apply the rule to.

8. Click **Next** to see a summary of the notification settings or **Finish** to save your settings and exit the wizard.

9. If you clicked **Next**, review your settings and then click **Finish** to save the notification rule and exit the wizard or **Prev** to edit your settings. You can also click the headings on the right of the wizard to go directly to a specific window.

   The notification will be added to the list of notifications in the Notification Rules window.

### Managing notification rules

In the **Notification Rules** window, you can edit, delete and enable/disable notification rules as described in the table below.

<table>
<thead>
<tr>
<th>To</th>
<th>Do This</th>
</tr>
</thead>
<tbody>
<tr>
<td>Delete a Rule</td>
<td>Select the rule and then click the <strong>Delete</strong> toolbar button. When prompted to confirm the deletion, click <strong>Yes</strong>.</td>
</tr>
<tr>
<td>Edit a Rule</td>
<td>Either double-click the rule you want to edit or select the rule and click the <strong>Edit</strong> toolbar button. Continue from <strong>Notifications (page 335)</strong>.</td>
</tr>
<tr>
<td>Disable a Rule</td>
<td>Select the rule you want to disable and then either click the <strong>Disable</strong> toolbar button or clear the check box in the <strong>Enabled</strong> column.</td>
</tr>
<tr>
<td>Enable a Rule</td>
<td>Select the rule you want to enable and then click the <strong>Enable</strong> toolbar button or select the check box in the <strong>Enabled</strong> column.</td>
</tr>
</tbody>
</table>

### Event IDs in Windows Event Log

The table below lists the Event IDs for Compose events in Windows Event Log.

- **Event ID**
- **Description**

<table>
<thead>
<tr>
<th>Event ID</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>261</td>
<td>The task ended with error</td>
</tr>
<tr>
<td>400</td>
<td>The task has started.</td>
</tr>
<tr>
<td>406</td>
<td>The task completed successfully.</td>
</tr>
</tbody>
</table>

*If a notification is set for several events, the event ID will be 0 for each of the events.*

**Windows Event Log IDs**

### Workflows

Workflows enable you to run tasks both sequentially and in parallel. You can either schedule workflows as described in **Scheduling tasks (page 334)** or run them manually using the **Run** toolbar button or Compose CLI.
You can create your own workflow and/or use the built-in workflow. The built-in workflow enables you to run all of your tasks as a single, end-to-end process. The built-in workflow appears in the Type column as "Default Workflow".

When you create your own workflow, you decide which tasks to include in the workflow and the order in which they will be run.

In this section:
- Adding and designing workflows (page 338)
- Validating workflows (page 340)
- Managing workflows (page 340)
- Running and monitoring workflows (page 341)

Adding and designing workflows
This section provides instructions for adding and creating workflows.

Adding a workflow

To add a workflow:

1. Switch to Monitor view by clicking the Monitor button in the top right of the Compose console.
2. Click the New Workflow toolbar button. The New Workflow window opens.
3. To create a workflow with all current tasks, select Create default workflow and then click OK. Otherwise, continue from Step 4 below. Separate workflows will be created for Full Load and Change Processing tasks. The default workflow cannot be edited and will appear as Default Workflow in the list of monitored tasks.

   Any tasks you create after adding the default workflow will not be automatically included in the default workflow. If you want to create a default workflow that includes newly added tasks, simply delete the existing default workflow and create another one in its place.

4. Specify a name for your workflow.
5. To create a workflow based on an existing workflow, select the Duplicate from check box and then select an existing workflow from the drop-down list.
6. Click OK to save your settings. The <workflow_name> window opens.
7. Continue from Designing a workflow below.

Designing a workflow
The workflow window is divided into two panes. The pane on the left (hereafter referred to as the Elements pane) is where you design your workflow and contains two default elements: Start and End.
The Elements pane contains gateways and tasks that you can use in your workflow. The following elements are available:

- **Tasks** - All existing Data Warehouse tasks, Data Mart tasks, and Command tasks.
- **Gateways** - There are two types of gateway: **Parallel Split** and **Synchronize**. Use the Parallel Split gateway to create parallel paths. This is useful, for example, if you want two or more tasks to run in parallel.
  Use the Synchronize gateway to merge parallel paths. The workflow waits for all the Tasks that precede the gateway to complete before continuing the flow.

**To design a workflow:**

1. Drag the desired workflow elements from the Elements pane to the pane on the left.
2. Arrange the elements in the order that you want them to run.
3. Connect the elements to each other by dragging the connector from the gray dot (that appears on the right of an element when you hover the mouse cursor over it) to the target element. When a blue outline appear around the target element, release the mouse button.
4. Optionally add error paths to the workflow. The workflow will follow the error path if a task encounters an error. For example, if an error occurs with one task, you may want to run another task in its place.
   To add an error path, hover your mouse cursor over the task element. A red dot will appears below the element. Drag the connector from the red dot to the target element, as shown below.

![Workflow Diagram](image)

*Connecting two error paths to the same task should be avoided as the workflow will fail if the task tries to run twice.*

**Continuing a workflow in the event of parallel task failure**

In a workflow, all task elements have an error port. This allows you to change the course of the workflow in the event of a task failure, as described in Step *Adding and designing workflows* (page 338) above. Similar to Task elements, the **Synchronize** gateway also has an error port which can be used to reroute the
workflow if any of the tasks between the **Parallel Split** and **Synchronize** gateways should fail.

By default, a workflow will end with an error if one or more parallel tasks do not complete successfully. However, in certain cases you may want the workflow to continue, even if one or more of the parallel tasks failed.

To do this, you need to connect the error port of the relevant task(s) directly to the **Synchronize** gateway. You can also design the workflow so that it follows the path leading from the **Synchronize** error port, instead of continuing its normal flow.

In the example below, the error port of the **MyCommandTask** is connected to the **Synchronize** gateway, meaning that even if **MyCommandTask** task fails, the workflow will continue. However, if the **MyCommandTask** task fails, the workflow will not proceed directly to the **End** element. Instead, it will follow the **Synchronize** gateway's error path to the **Source** task.

### Validating workflows

It is strongly recommended to validate your workflow before running it. This will prevent errors from occurring during runtime due to an invalid workflow.

Workflow rules include:

- All elements must be connected to each other
- A workflow must contain Start and End elements and at least one task.
- A workflow cannot contain a Parallel Split gateway without a Synchronize gateway and vice versa.
- Storage Zone tasks that update the same tables cannot run in parallel.
- A workflow cannot contain a Parallel Split gateway without a Synchronize gateway and vice versa.
- The execution order of elements must be sequential and not cyclic. For example an element cannot loop back to an element that precedes it the execution order.

To validate your workflow:

- Click the **Validate Flow** toolbar button.

If the workflow is valid, a "<workflow_name> is valid." message will be appear at the top of the window. If the workflow is not valid, a message describing the problems will appear instead.

### Managing workflows

The table below describes the options available for managing workflows.
To | Do This
--- | ---
Delete a Workflow | In Monitor view, select the workflow in the Task column and then click the Delete Workflow toolbar button.
Edit a Workflow | In Monitor view, either double-click the workflow you want to edit or select the workflow and click the Open toolbar button. Continue from Adding and designing workflows (page 338).
Delete an element in workflow | Either right-click the element and select Delete or select the element and then click the Delete toolbar button.
Reset the workflow view | Click the reset button to the right of the slider at the top of the window.
Zoom in to/zoom out of the workflow | Move the slider at the top of the window to the left or right as required.

Running and monitoring workflows

You can either schedule workflows as described in Scheduling tasks (page 334) or run them manually using the Run toolbar button. The Run toolbar button appears both in the main Monitor view and in the workflow design window. Note that when you run a workflow from the workflow design window, a new Monitor tab is added to the window and the view automatically switches to the Monitor tab.

You can monitor the workflow either in the Monitor tab or in the Progress Status tab. During runtime, the workflow elements fill with blue providing a graphic indication of progress. If a task encounters an error, the task element will appear with red fill instead of blue.

Monitoring and controlling Replicate tasks

Before you can create a Compose project, you need to define a Replicate task that replicates the relevant source tables from the source database to the Landing Zone. You can define a different task for each project or the same task can serve several projects. You can also define multiple tasks for a single project. The tasks can either reside on the same Replicate server or on several Replicate servers distributed throughout your organization.

Monitoring and controlling Replicate tasks from within Compose involves the following steps:

- **Step 1:** Configure Qlik Compose to connect to the Qlik Replicate machine(s) as described in Replicate Server settings (page 347).
- **Step 2:** Add the Replicate task name to the source Landing Zone settings as described in Defining Landing Zones connections (page 292).
- **Step 3:** Monitor and control the Replicate task as described below.

The image Replicate Task in the Compose Monitor (page 342) shows how the Replicate task appears in the Compose Monitor. You can stop and start the Replicate task using the Abort and Run toolbar buttons.
If a task is stopped from within Replicate, the task status in Compose for Data Lakes will be "Completed" instead of "Aborted".

You can also define notifications for the task and add the task to a workflow. For more information, see Notifications (page 335) and Workflows (page 337) respectively.

The monitor provides various information about the task. For details, see Viewing information in the monitor (page 332).

Replicate Task in the Compose Monitor
Managing Compose

Qlik Compose management options can be accessed from the Management menu located at the top of the Compose main page.

In this section:
- License settings (page 343)
- Logging settings (page 344)
- Mail server settings (page 347)
- Running tasks on a remote Compose server (page 347)
- Replicate Server settings (page 347)
- User permissions (page 348)
- Audit trails (page 355)

7.1 License settings

You need to register a valid license in order to use Qlik Compose. The license file contains details such as the product expiration date, the date the license was issued, which source databases can be used, and so on.

License enforcement

The license is enforced only when trying to generate, run, or schedule a task (via the web console or API ). Other operations such as Test Connection may also fail if you do not have an appropriate license.

Registering a license

This section describes how to register your Compose license. You can register the license either using the console or using a command line.

To register a license using the console:

1. Copy the license file to the computer on which Compose is installed or to any computer in your network that can be accessed from the Compose computer.
2. Click Load and browse to find and select the license file. The license text is displayed in the window. Check to be sure that the details are correct.
3. Click Register License to register the license. A message indicating the license was registered successfully is displayed.

To register a license using the command line:

1. Open a command prompt and change the working directory to:
   
   PRODUCT_DIR\bin
   

7 Managing Compose

Qlik Compose management options can be accessed from the Management menu located at the top of the Compose main page.

In this section:
- License settings (page 343)
- Logging settings (page 344)
- Mail server settings (page 347)
- Running tasks on a remote Compose server (page 347)
- Replicate Server settings (page 347)
- User permissions (page 348)
- Audit trails (page 355)
2. Run the following command to connect to the server:
   `ComposeCli.exe connect`

3. Issue the following command:
   `ComposeCli.exe register_license --infile=license_text`
   where:
   - `infile` is the full path to the Qlik Compose license file.
   - `license_text` is a string in JSON format. When specifying a JSON string any quote symbols should be escaped using a backslash (`\`).

   **Example:**
   ```bash
   ComposeCLI.exe register_license --infile "c:\Admin\Temp\lic.txt"
   ``

   If the license is registered successfully, the following message will be displayed:
   `Compose control program completed successfully.`
   Otherwise, an appropriate error message will be displayed.

7.2 Viewing a license

You can view the license information in the Qlik Compose Console at any time.

**To view the license information:**

1. From the Management menu, select License|View License.
2. The License window opens. All of the license information is displayed in the License window.

7.3 Logging settings

You can set the server logging level, configure automatic roll over and cleanup, and view and download log files.

Setting the logging level

The logging level determines what type of information is written to the log files. The log files provide information about Qlik Compose Server and Qlik Compose Agent processes.

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

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 files. However, if you select Info, informational messages, warnings, and error messages will be included. SelectingVerbose writes all possible messages to the log.

To set the server logging level:

2. To set a global logging level, move the top slider to the desired logging level. All of the sliders for the individual modules move to the same level that you set in the main slider.
3. To set a logging level for individual Compose components, select a module and then move its slider to the desired logging level.
4. Click OK to save your changes and close the Log Management window.

To set the Qlik Compose Agent level:

2. Select the Qlik Compose Agent log tab, and then move the slider to the desired logging level.
3. Click OK to save your changes and close the Log Management window.

Changes to the logging level take place immediately. There is no need to restart the Qlik Compose service.

Setting automatic roll over and cleanup

You can define when log files should be automatically rolled over as well as how many log files to keep. Rolling over log files keeps any single log file from becoming too large and provides an easy way to identify files that are no longer being used so that an automated script can clean the logging directory. Automatic deletion of old log files ensures that the logs do not take up too much disk space when there is a lot of activity in the system.

To set the log file roll over and cleanup options:

1. From the Management menu, select Logs|Log Management. The Log Management window opens.
2. Select the Log Settings tab.
3. The following options are available:
   - Enable automatic roll over: Select this check box to determine the maximum size a log file can reach before it is rolled over. The current log file is called Compose.log and saved (older) log files are called Compose_xxxxxxxxxxxx.log where xxxxxxxxxxx represents a 12-digit timestamp.
7 Managing Compose

- **Roll over the log if the log file is larger than (MB):** Use the counter or type in the maximum amount of megabytes for a specific log file. 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 default value is 10 megabytes.

> The scheduled job that checks the log size runs every five minutes. Consequently, the actual size of the log when rolled over might be larger than specified.

- **Enable automatic cleanup:** Select this check box to define the maximum number of log files to keep.

  - **Maximum number of log files to keep:** Use the counter or type in the maximum number of log files to keep. When the number of log files reaches the specified maximum, Compose will delete the oldest log file whenever a new log file is created, thereby ensuring the number of log files never exceeds the set limit. The default is 45.

4. Click **OK** to save your settings and close the Log Management window.

**Viewing and downloading Compose log files**

This section explains how to view and download Compose log files.

> The logs are in four different locations:

- `<product_dir>/data/logs - Server log file`
- `<product_dir>/java/data/logs - Compose Agent log file`
- `<product_dir>/data/projects/<project_name>/logs - project logs, workflow logs, and command task logs`

**To view Compose log files:**

1. From the **Management** menu, select **Logs|View Logs**. The Log File Viewer opens.
2. Select the log file you want to view from the list in the **Log Files** pane.
   - 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 display the full message of the selected row.
3. Browse through the log file using the scroll bar on the right and the navigation buttons at the top of the window.
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 will be highlighted blue.

**To download Compose log files:**

1. From the **Management** menu, select **Logs|View Logs**. The Log File Viewer opens.
2. From the list in the **Log Files** pane, select the log file you want to download.
3. Click the **Download Log File** button in the top right of the window. The log file is downloaded.
7.4 Mail server settings

The mail parameters define the mail server used to send notifications.

To configure the mail server settings:

1. From the Management menu, select Mail Server Settings. The Mail Settings window opens.
2. Configure the settings as follows:
   - Mail server: Specify the outgoing mail server that will be used to send Qlik Compose notifications, for example, smtp.example.com.
   - Port: Enter the mail server port number. The default value is 25.
   - Use secure email (SMTPS): Select this to connect to the mail server using TLS.
   - Anonymous login: Enable this to allow Qlik Compose to access the mail server without having to provide any user credentials.
   - User name: Specify the user name for the account that will be used to send notifications.
   - Password: Specify the password for the account that will be used to send notifications.
   - Sender email address: Enter the email address that sends the email notifications. This is the address that appears in the From field of the email notification.
   - Send Test Mail: You this option to validate your mail server settings. Click Send Test Mail to open the Send Test Email window. In the Email address for test email, enter the email address to which you want the test email to be sent and then click Send.
3. Click OK to save your settings and close the Mail Settings window.

7.5 Running tasks on a remote Compose server

You can run Compose tasks either locally (the default) or on a remote Compose server.

To run tasks on a remote server:

1. From the Management menu in the projects view, select Compose Agent Settings.
2. In the Compose Agent Settings window, select Remote server and provide the required connection details.
3. Click OK to save your settings.

7.6 Replicate Server settings

Before you can create a Compose project, you need to define at least one Replicate task that replicates the relevant source tables from the source database to the Landing Zone.

If you want to monitor the Replicate tasks, you need to provide the information that Compose needs in order to establish a connection to the Replicate Server on which the tasks are running. After providing this information, you will then be able to associate a source Landing Zone with a specific Replicate task.
To configure the Replicate Server connection settings:

1. Open the Manage Replicate Servers window using any of the following methods:
   - From the Management drop-down menu in the main toolbar, select Manage Replicate Servers.
   - In the New Data Source window, click the Replicate Server Settings link below the Associate with Replicate task field.
   The Manage Replicate Servers window opens.
2. Click Add Replicate Server.
   The Add Server window opens.
3. Enter the following information:
   - Name: A display name for the server.
   - Description: (Optional) A description for the server.
   - Host: The IP address or host name of the Qlik Replicate machine.
     When Replicate Server is installed on Linux, enter the IP address of the Windows machine on which the Replicate UI Server is running.
   - Port: Optionally, change the default port (443). You should only change the default port if you are certain that a different SSL port is being used.
   - User Name and Password: Your credentials for logging in to the Qlik Replicate machine.
     When Replicate Server is installed on Linux, enter the user name and password for the Windows machine on which the Replicate UI Server is running.
   - Get metadata timeout - The time to wait when discovering a task’s source database or refreshing the metadata cache before returning a timeout error.
   - Get task timeout - The time to wait when starting a Replicate task before returning a timeout error.
     In environments with complex networks, operations related to Replicate may exceed the default timeout limit. If you experience frequent timeouts starting tasks, discovering a task’s source database, or refreshing the metadata cache, increasing these values may help.
4. Click Test Connection and then click OK if the connection is successfully verified.
   The server is added to the Manage Replicate Servers window. Click Close to close the window.

7.7 User permissions

Security roles allow you to grant Qlik Compose users different roles according to the tasks you want them to perform. Qlik Compose comes with the following predefined security roles: Admin, Designer, Operator and Viewer. Each role has its own set of permissions, as described in Default user permissions according to role (page 349).
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 Qlik Compose is an Admin. You can also fine-tune access control per user or group. For more information, see *Granular access control* (page 350)

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 Qlik Compose.

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 is added to an existing group automatically acquires the security role granted to that group.

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 352).

<table>
<thead>
<tr>
<th>Predefined user permission roles</th>
</tr>
</thead>
<tbody>
<tr>
<td>Role</td>
</tr>
<tr>
<td>Administrator</td>
</tr>
<tr>
<td>Designer</td>
</tr>
<tr>
<td>Operator</td>
</tr>
<tr>
<td>Viewer</td>
</tr>
</tbody>
</table>

### Default user permissions according to role

In the Qlik Compose Console, the menus, buttons, and options can be accessed only by users who have the relevant permissions. For example:

- **The Project** view is available to all roles, but Designers only have read-access to user permissions, and operators cannot add projects - they can only view the different settings but not edit them. Viewers cannot edit settings, add, edit, or delete a project, or register a license.

- **The Model** view for the Data Warehouse is available to all roles, but only Designers can create and manage the model, import entities and mappings from other projects (including models created in ERwin), manage global mappings, validate, define reusable transformations, add Date and Time entities for the model, and so on.

The following table lists the permissions granted to each of the predefined security roles:

<table>
<thead>
<tr>
<th>Permission</th>
<th>Admin</th>
<th>Designer</th>
<th>Operator</th>
<th>Viewer</th>
</tr>
</thead>
<tbody>
<tr>
<td>Projects: View projects and logs, generate documentation</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
</tr>
</tbody>
</table>
### Managing Compose

<table>
<thead>
<tr>
<th>Permission</th>
<th>Admin</th>
<th>Designer</th>
<th>Operator</th>
<th>Viewer</th>
</tr>
</thead>
<tbody>
<tr>
<td>Projects: Define and manage command tasks</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Projects: Create, design, reset, define settings for, control versions, commit, revert, delete, create deployment package</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
<td>No</td>
</tr>
<tr>
<td>View databases and model</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Model: Add, edit, delete, discover, import from other tools (e.g. ERwin), validate</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
<td>No</td>
</tr>
<tr>
<td>Data Warehouse: View data and logs, tasks, commands and mapping</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Data Warehouse: Create, edit, delete source and target Data Warehouse databases</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
<td>No</td>
</tr>
<tr>
<td>Data Warehouse: Manage settings, populate tasks, cleaning and validation rules</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
<td>No</td>
</tr>
<tr>
<td>Data marts: View details and logs</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Populate data marts</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Data marts: Create, edit, delete, edit expressions and filters, import and add dimensions, generate tasks</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
<td>No</td>
</tr>
<tr>
<td>Monitor tasks (in Monitor view)</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Workflow operations: Create, edit, populate, notification rules, run Replicate and command tasks</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Perform runtime operations such as starting and aborting tasks.</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Define table creation modifiers</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
<td>No</td>
</tr>
<tr>
<td>Manage Compose (e.g. license registration, email settings, and so on)</td>
<td>Yes</td>
<td>No</td>
<td>No</td>
<td>No</td>
</tr>
</tbody>
</table>

*Applies only if the user is an Admin at the Compose level.*

#### Granular access control

For each user, Qlik Compose 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 the
decentralization of control, effectively preventing the same user from, for example, designing the model and managing the mappings. As such, granular access control lets you create a buffer between those who can create and design models and those who can create and run the mappings.

Qlik Compose 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.

User permissions can be assigned to individual Data Warehouse projects as well as across all projects. The following hierarchy is in place, whereby:

- **Compose** User Permissions are applied globally. Changes to Compose permissions will affect any level that inherits those permissions. At Compose root level, users must have at least Viewer permissions.

  > Only Admin users at the Compose level can perform logging actions, such as, changing the logging level and rolling over logs.

- **All Projects** User Permissions apply to all projects. When inheritance is enabled (the default), permissions will be inherited from the “Compose” root level.

  > A user that is assigned All Projects User Permissions but not Compose User Permissions is not authorized to log in to Compose.

- **Project** User Permissions apply to a specific project. When inheritance is enabled (the default), permissions will be inherited from the “All Projects” level.

  > Currently supported with Data Warehouse projects only.

- **Model** User Permissions apply to the model unless overridden at any of the lower levels. When inheritance is enabled (the default), permissions will be inherited from the “Project” level.

  - Not applicable to Data Lake projects
  - If user permissions are other than None at the Project level, in the Model level the user must have at least Viewer permissions.

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:
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.

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.

In the User Permissions window, inheritance is indicated by a check mark 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. Qlik Compose 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 352).

Managing user permissions

This section explains how to access user permissions at different levels in the hierarchy, 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 350) and Inheritance and overrides (page 351).

To access user permissions at the Compose level

- In the Qlik Compose Console, from the Management menu, select User Permissions.

By default, the User Permissions window opens at Console root level, displaying the currently assigned user role permissions for each defined user/group. These permissions apply globally unless they are overridden at any of the lower levels.
Changes to Compose permissions will affect any level inheriting those permissions.

To access user permissions at the All Projects level

- In the **User Permissions** window, select the **All Projects** tab.

The **All Projects User Permissions** window displays the currently assigned user role permissions for each defined user/group. These permissions apply to all projects unless they are overridden at any of the lower levels.

> When inheritance is enabled, permissions will be inherited from the Compose root level.

To access user permissions for a specific project:

- In the Qlik Compose Console, select the required project, and then select **User Permissions** from the context menu.

The **Project User Permissions** window shows the user role permissions that apply to the specific project `{project name}` for each defined user/group. These permissions apply to the specific project unless overridden at any of the lower levels.

> When inheritance is enabled, permissions will be inherited from the **All Projects** level.

To access user permissions for a Model:

- In the **User Permissions for project: `{project name}`** window, click the **Model** tab. The **Model User Permissions** window shows the user role permissions that apply to the specific project model for each defined user/group.

> When inheritance is enabled, these permissions will be inherited from the **Project** level.

To edit user permissions:

1. In the **User Permissions** window, adjust the permission slider for a user or group as required.

   > Adjusting the slider stops inheritance from the parent object.

2. Click **Save** or **OK** to accept the changes, or **Discard Changes** or **Cancel** to undo them.

To add a user or group:

1. In the **User Permissions** window, click **Add**.

2. In the **Add User/Group** window, 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)

4. Click **OK** to add the user/group and close the window.

5. Click **Save** or **OK** to accept the changes, or **Discard Changes** or **Cancel** to undo them.
To remove a user or group:
1. In the User Permissions window, select the user or group you want to remove.
2. Click Remove.
3. When prompted, click Yes to confirm.
4. Click Save or OK to accept the changes, or Discard Changes or Cancel to undo them.

To disable inheritance:
1. In the User Permissions window, click Disable Inheritance.
   
   *This option disconnects the entire authorization level from the parent level.*

2. In the Disable Inheritance window, 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 permissions, all users and groups disappear from the list.
4. Click Save or OK to accept the changes, or Discard Changes or Cancel to undo them.

To enable inheritance:
1. In the User Permissions window, click Enable Inheritance.
   
   *This option enables inheritance for all users and groups on this level.*

2. In the Enable Inheritance window, 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 window, 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 window, 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`)

2. Click Get Effective Permissions. The effective permissions for the user you entered appear below the button.

## 7.8 Audit trails

The information provided in an Audit Trail can be leveraged for user accountability, reconstruction of events, intrusion detection, and other operational issues. As such, Audit Trails are an indispensable tool for regulatory compliance (e.g. SOX).

For operations performed by users with Operator privileges or later, the Compose Audit Trail shows which user performed the operation, when it was performed, and on which objects.

Compose retains audit files for two weeks or until they reach a total size of 500 MB (50 files). You can change these settings through the command line interface (CLI) as described in Exporting Audit Trail files (page 356) below.

Audit Trail files are located in the following folder:

`<Installation_Directory>\data\AuditTrail\audit_service`

You can also export an audit trail file for a specific time range, as described in Exporting Audit Trail files (page 356).

### Audit trail information

Audit Trail files provide all or some of the following information:

- **Timestamp** - The time when the row was inserted into the Audit Trail.
- **User** - The user that performed the operation.
- **Node** - The IP of the server on which the operation was performed.
- **Requested Action** - The API method/function that was called.
- **Required Permission** - The minimum role of the user that can perform the operation.
- **Effective Permission** - The actual role of the user that performed the operation.
- **Security Result** - Whether the user is allowed to perform the operation.
- **Action Result** - The completion status of the operation (success of failure).
- **Error Message** - The error message if the operation failed.
- **Task** - The name of the task where relevant.
- **Notification** - The notification defined for the operation (if defined).
Managing Compose

- **Payload** - A URL. To view payload information, simply copy the link from the **Payload** column and paste it into your browser's address bar. Payloads for some operations (e.g., RegisterLicense) contain sensitive information and need to be decoded. For information on decoding payloads, see *Decoding an encoded payload* (page 357).

- **Project Name** - The name of the Compose project.

---

*Audit Trail files are compressed and tamper-protected.*

### Exporting Audit Trail files

You can export an audit trail file with a record of activity for a specific time range. In Compose, there are two ways of doing this:

- Using the management console
- Using the CLI

---

*You can also export audit trails using the ExportAuditTrail API method. For further information, see the Qlik Enterprise Manager Help and API Guide.*

### Exporting an Audit Trail file via the management console

You can use the Compose management console to export the audit trail as a CSV file.

To do this:

1. From the **Management** drop-down menu, select **Audit Trail**. The **Audit Trail** window opens.
2. From the **Time Range** drop-down list, select the desired time range. If you select **Custom**, set **From** and **To** values as well.
3. Click **Generate**.

Depending on your browser settings, you will either be prompted for a download location or the file will be downloaded automatically to your preferred location.

### Exporting an Audit Trail file via the CLI

You can use the Compose CLI to export the audit trail as a JSON file.

To do this:

1. Open a command prompt as Admin and switch the path to `<COMPOSE_INSTALL_DIR>/bin`.
2. Establish a connection to the Compose server by running the following command:

   ```
   composeCli.exe connect
   ```

3. To export the audit trail, run the following command:

   ```
   composeCtl.exe generate_audit_trail --start_timestamp timestamp [--end_timestamp timestamp] --outfile full_path
   ```
Example:

```
ComposeCtl.exe generate_audit_trail --start_timestamp 2020-06-30T16:15:00Z --end_timestamp 2020-07-14T16:15:00Z --outfile "C:\compose audit trails\audit.json"
```

Where:
- `--start_timestamp` is the date and time from which you want the audit trail to start, in UTC format.
- `--end_timestamp` is the date and time on which you want the audit trail to end, in UTC format. This is optional. When not specified, the file will end at the latest audit trail record.
- `--outfile` is the full path to the output file. If the path contains spaces, it should be enclosed in quotation marks.

Configuring Audit Trail size and retention

To do this:

1. Open a command prompt and change the working directory to:
   ```
   <COMPOSE_INSTALL_DIR>\bin
   
   Default: C:\Program Files\Qlik\Compose\bin
   ```
2. Run the following command:
   ```
   ComposeCtl.exe connect
   
   The following message should be displayed:
   ComposeControl Program completed successfully.
   ```
3. Run the following command:
   ```
   ComposeCtl.exe audit_trail control --age weeks --size megabytes
   
   Where:
   - `weeks` is the number of weeks to retain the audit trail file (default 2 weeks).
   - `megabytes` is the maximum size of the audit file to retain (default 500 MB).

Decoding an encoded payload

Some audit records (e.g. RegisterLicenses) may contain an encoded 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.
   ```
   M N O P Q R S T U V W X Y Z AA AB AC
   Payload
   [https://skQzgA.aq.rr/attunitycompose_datawarehouses/rest/audit_trail?action=ret_json&payload_id=6370154175089332&hex=66afccc3b948d0ecb9b65279160]
   ```
2. Copy the URL into your browser's address bar and press [Enter]. A byte array will be displayed.
3. Copy the byte array into a Base64 decoder and decode it.
8. Setting up Compose on a Windows HA cluster

This topic describes how to set up Compose in a Windows Server High Availability Cluster environment. For instructions on how to set up a Windows clustering environment, refer to the Microsoft Help.

When building failover cluster solutions with Compose 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 Compose DATA folder. Using NFS or SMB-based storage is not supported due to 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:

- **Step 1: Installing Compose in the cluster (page 358)**
- **Step 2: Adding the Compose service (page 359)**
- **Step 3: Defining the service dependencies (page 360)**
- **Step 4: Defining the URL for the cluster (page 360)**
- **Upgrading Compose on the cluster (page 361)**

### 8.1 Step 1: Installing Compose in the cluster

This topic describes how to install Compose in a high availability cluster environment.

**Preparation**

1. Allocate two shared folders for Compose: one for the Compose server and the other for the Compose agent.
   
The setup instructions below assume that the Compose data folder is `F:\server-data` and the Compose Agent data folder is `F:\agent-data`.
2. Generate a 32 character random master key by running the following command from `<PRODUCT_DIR>\bin`:
   
   ```
   ComposeCtl.exe utils genpassword
   ``
   
The setup instructions below assume that your key is `WdAHWEwXSvwxFdctlTvVF8SXPbMrFx`

**Primary node setup**

1. Install Compose.
2. Stop the Compose service.
3. Edit the service executable path as follows:
   
   ```
   SC CONFIG ComposebinPath="<PRODUCT_DIR>\bin\ComposeCtl.exe -d F:\server-data service run"
   ```
4. Run the following command from `<PRODUCT_DIR>\bin`:
   ```
   ComposeCtl.exe -d "F:\server-data" setup install
   ```

5. Run the following command from `<PRODUCT_DIR>\bin`:
   ```
   ComposeCtl.exe -d "F:\server-data" masterukey set -p WdAHWEwXSvwxDFetcl7TVVFfSXpMrFx
   ```

6. Edit `<PRODUCT_DIR>\java\bin\acjs.bat` and immediately below the line with `SET JAVA_LIB_PATH`, add the following:
   ```
   set AT_DATA=-d F:\agent-data
   ```

7. Start the Compose service and then stop it. This will create the java repository.

   **This step should be performed on the primary node only.**

8. Run the following command from `<PRODUCT_DIR>\java\bin`:
   ```
   acjs.bat masterukey set WdAHWEwXSvwxDFetcl7TVVFfSXpMrFx
   ```

9. In the Cluster Manager, move to the secondary node.

**Secondary node setup**

Perform steps 1-6 and step 8 (step 7 is not required) of the the Primary node setup (page 358) steps on the secondary node and then start the Compose service.

### 8.2 Step 2: Adding the Compose service

After installing Compose in the cluster, you need to add the Compose service as a resource to the service (Windows Server 2008 Cluster) or role (Windows Server 2012\2016 Cluster).

**To add the Compose service:**

1. Do one of the following (according to your Windows Server version):
   - **Windows Server 2008 Cluster:** In the left pane of the Failover Cluster Manager, right-click the service and point to Add a resource. Then select Generic Service.
   - **Windows Server 2012\2016 Cluster:** In the left pane of the Failover Cluster Manager, select Roles. The available roles will be listed in the right pane of the console. Right-click the role you are working with and point to Add a resource. Then select Generic Service.

2. In the Select Service screen of the New Resource wizard, select Compose from the list.

3. Click Next and follow the directions in the wizard to create the resource. For information on how to use this wizard, see the Microsoft online help.

   **Compose must be installed on the computer where you defined the service in order for the service to be available in the list.**
8.3  Step 3: Defining the service dependencies

You need to define dependencies for the Compose service that will enable the Storage and Network names to start up before the service. If these resources do not start before the service, Compose will not be able to start as it will be searching for the data location.

To define the service dependencies:

1. Do one of the following (according to your Windows Server version):
   - **Windows Server 2008 Cluster**: In the left pane of the Failover Cluster Manager, select the Compose service. The properties for this service are displayed in the center pane.
   - **Windows Server 2012\2016 Cluster**: In the left pane of the Failover Cluster Manager console, select Roles. From the list of available roles in the right pane of the console, select the role you are working with and then, in the bottom right pane, select the Resource tab. From the list of the available roles, select Compose.

2. Do one of the following (according to your Windows Server version):
   - **Windows Server 2012\2016 Cluster**: Right-click the Compose role and select Properties.

3. In the Compose Properties window that opens, select the Dependencies tab.

4. Click Insert. A new line is added to the Resource list.

5. In the Resource column, click the arrow and select the Compose Data storage resource from the list.

6. Click Insert and add the Network Name resource (it should have the same name as the cluster).

7. Start the Service using the Failover Cluster Manager and access the console using the Network name.

8. Register the license. The license should contain all host names of the cluster.

To open the Compose Console, it is recommended to use an address that includes the name or IP address of the cluster machine (as opposed to the specific node name).

Example:

https://cluster_name_ip/qlikcompose/2021.2.0.34/#

8.4  Step 4: Defining the URL for the cluster

By default, the Compose service generates the URL when it starts, according to the hostname of the machine on which Compose is installed.

In a cluster environment, this is not good practice because the URL will change each time the cluster is rolled over. To resolve this issue, you need to set the cluster name as the Compose URL.
To set the cluster name as the Compose URL:

1. In the left pane of the Failover Cluster Manager, select **Nodes**. The right pane of the Console displays a list of cluster nodes.
2. Select a node to see the cluster name. This is the name you want to set (for example: `cluster_network_1`).

   The cluster name must be registered in DNS, before you can set it.

3. Run the following command from the primary node:
   ```
   <PRODUCT_DIR>\bin>ComposeCtl.exe -d <COMPOSE_DATA_FOLDER> configuration set --address <CLUSTER_NAME>
   ``
   The host configuration will be updated.

4. Restart the Compose service for the changes to take effect.
5. To make sure Compose is now using the correct URL, use the `\service.url` shortcut to check the cluster name in the service Properties.
6. Try to open Compose from a remote browser using `\service.url`.

### 8.5 Upgrading Compose on the cluster

To upgrade Compose on a Windows Server High Availability Cluster:

1. Make sure you are on the primary node.
2. In the left pane of the Failover Cluster Manager, select **Roles**. From the list of available roles in the right pane, right-click the Compose role you are working with and change it to offline.
3. Run the standard upgrade procedure.

   Make sure the Compose role is offline, as the upgrade should bring the services online.

4. As the upgrade process overrides the `acjs.bat` file, when the upgrade completes, add the following row to the `\javalib\acjs.bat` file:
   ```
   SET AT_DATA=-d <agent data path>
   ``

   If the above string already exists in `acjs.bat`, you can skip this step.

5. Bring the Compose role back online and make sure there are no connection errors.
6. Upgrade the projects by running the following command on the primary node only:
   ```
   ComposeCtl.exe -d <server data path> setup postupdate
   ```
7. Repeat steps 2-5 on the secondary node.
A  Impact of DST change on Qlik Compose

This topic describes how Qlik Compose is affected by Daylight Saving Time (DST) and provides guidelines for handling changes brought about by DST.

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, 01:00AM becomes 02:00AM). This DST change does not impact Qlik Compose as 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, 02:00AM becomes 01:00AM). 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 Compose is stopped.

There are two places where DST may have an effect:

- 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.

- 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).

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 Compose, other components are also affected including:

- The source endpoint system
- The target endpoint system
- The local operating system
- The task design (specifically using timestamp based variables)

*The Scheduler is not adjusted to take into account daylight saving time (DST). For example, a daily job which was scheduled to run at 11 PM should be rescheduled to run at 11 PM after DST comes into effect.*

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.
B Support matrix

In addition to listing the platforms on which Qlik Compose can be installed, this topic also specifies which source and target database versions can be used in a Qlik Compose task.

B.1 Supported Windows platforms

Qlik Compose can be installed on any of the following Windows platforms:

- Windows Server 2012 (64-bit)
- Windows Server 2012 R2 (64-bit)
- Windows Server 2016 (64-bit)
- Windows Server 2019 (64-bit)

B.2 Supported browsers

The Qlik Compose Web UI supports the following browsers:

- Microsoft Internet Explorer version 11 or later
- Microsoft Edge (with automatic updates turned on)
- Mozilla Firefox (with automatic updates turned on)
- Google Chrome (with automatic updates turned on)

B.3 Supported Qlik Replicate and Enterprise Manager versions

Qlik Replicate is required for landing data into the data warehouse or storage while Qlik Enterprise Manager offers a convenient way of monitoring and controlling Compose tasks running on different servers. This section lists the supported versions for each of these products.

Compose August 2021 Initial Release

Compose August 2021 Initial Release is compatible with the following Replicate and Enterprise Manager product versions:

- **Qlik Replicate** - Qlik Compose is compatible with Replicate November 2020 latest service release, and Replicate May 2021 latest service release.
- **Enterprise Manager** - Qlik Compose is compatible with Enterprise Manager May 2021 latest service release.

Compose August 2021 Service Packs 01 and 02

Compose August 2021 Service Packs 01 and 02 are compatible with the following Replicate and Enterprise Manager product versions:
**B.4 Supported Databases for Data Warehouse Projects**

**Supported data sources**

Any data source supported by Qlik Replicate can be used as a data source in Qlik Compose. When using a Qlik Replicate data source, discovery needs to be performed on the landing zone. Replicate data sources (endpoints) that can be discovered directly from Qlik Compose are described in the table below.

For more information on discovering data sources, see *Discovering the Source Database or Landing Zone (page 142).*

<table>
<thead>
<tr>
<th>Data Source (Endpoint)</th>
<th>Version</th>
</tr>
</thead>
<tbody>
<tr>
<td>MySQL</td>
<td>5.x and 8.x</td>
</tr>
<tr>
<td>Oracle</td>
<td>11.x, 12.x, 18.x, and 19.x</td>
</tr>
<tr>
<td><strong>Note:</strong> All Oracle editions are supported.</td>
<td></td>
</tr>
<tr>
<td>IBM DB2 for LUW</td>
<td>9.x, 10.x, and 11.x</td>
</tr>
</tbody>
</table>

**Supported data warehouses**

The table below lists the data warehouse versions supported in a Data Warehouse project.

<table>
<thead>
<tr>
<th>Data Warehouse</th>
<th>Version</th>
</tr>
</thead>
<tbody>
<tr>
<td>Microsoft Azure SQL Database</td>
<td>Same as Microsoft SQL Server.</td>
</tr>
<tr>
<td>Microsoft Azure SQL Managed Instance</td>
<td>Same as Microsoft SQL Server.</td>
</tr>
<tr>
<td>Oracle</td>
<td>11.x, 12.x, 18.x, and 19.x</td>
</tr>
<tr>
<td><strong>All Oracle editions are supported.</strong></td>
<td></td>
</tr>
</tbody>
</table>
| Amazon Redshift                         | N/A

- **Qlik Replicate** - Qlik Compose is compatible with Replicate May 2021 latest service release and Replicate November 2021 (and its subsequent service releases).
- **Enterprise Manager** - Qlik Compose is compatible with Enterprise Manager November 2021 (and its subsequent service releases) only.
## Support matrix

<table>
<thead>
<tr>
<th>Data Warehouse</th>
<th>Version</th>
</tr>
</thead>
<tbody>
<tr>
<td>Microsoft Azure Synapse Analytics (formerly known as Microsoft Azure SQL Server Data Warehouse)</td>
<td>N/A</td>
</tr>
<tr>
<td>Snowflake</td>
<td>N/A</td>
</tr>
<tr>
<td>Google Cloud BigQuery</td>
<td>N/A</td>
</tr>
</tbody>
</table>
B.5  Supported databases for Data Lake projects

The table below lists the supported databases for Data Lake projects.

- For all databases, fully binary compatible versions are also supported.
- All major versions and selected minor versions are certified for use with Compose.
- For information about supported drivers, see Prerequisites (page 257).

<table>
<thead>
<tr>
<th>Hive Distribution</th>
<th>Version</th>
</tr>
</thead>
<tbody>
<tr>
<td>Amazon EMR</td>
<td>6.x</td>
</tr>
<tr>
<td>Cloudera</td>
<td>7.x</td>
</tr>
<tr>
<td>Microsoft Azure HDInsight</td>
<td>4.x</td>
</tr>
<tr>
<td>Google Dataproc (non ACID)</td>
<td>2.0</td>
</tr>
<tr>
<td>Databricks on AWS</td>
<td>7.1 - 7.3 (LTS)</td>
</tr>
<tr>
<td>Databricks on Azure</td>
<td>7.1 - 7.3 (LTS)</td>
</tr>
<tr>
<td>Databricks on Google Cloud</td>
<td>8.x</td>
</tr>
</tbody>
</table>
C Cron format and examples

Cron expressions can be used to schedule a Compose task. This appendix describes the Cron format used in Compose (Quartz), provides a description of the special characters that can be used in an expression and ends with some examples of Cron usage.

In this appendix:

C.1 Cron format

A cron expression is a string comprised of five fields separated by a white space. Fields can contain any of the allowed values, along with various combinations of the allowed special characters for that field. The fields are described in the table below.

<table>
<thead>
<tr>
<th>Field Name</th>
<th>Mandatory</th>
<th>Allowed Values</th>
<th>Allowed Special Characters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Seconds</td>
<td>✔️</td>
<td>0–59</td>
<td>, * /</td>
</tr>
<tr>
<td>Minutes</td>
<td>✔️</td>
<td>0–59</td>
<td>, * /</td>
</tr>
<tr>
<td>Hours</td>
<td>✔️</td>
<td>0–23</td>
<td>, * /</td>
</tr>
<tr>
<td>Day of month</td>
<td>✔️</td>
<td>1–31</td>
<td>, * ? / L W</td>
</tr>
<tr>
<td>Month</td>
<td>✔️</td>
<td>1–12 or JAN–DEC</td>
<td>, * /</td>
</tr>
<tr>
<td>Days of week</td>
<td>✔️</td>
<td>1–7 or SUN–SAT</td>
<td>, * ? / L #</td>
</tr>
</tbody>
</table>

C.2 Special characters

The following special characters are supported:

- * ("all values") Used to select all values within a field. For example, "*" in the minute field means "every minute".
- ? ("no specific value") Useful when you need to specify something in one of the two fields in which the character is allowed, but not the other. For example, if I want my task to run on a particular day of the month (say, the 10th), but don't care what day of the week that happens to be, I would put "10" in the day-of-month field, and "?" in the day-of-week field. See the examples below for clarification.
- - Used to specify ranges. For example, "10–12" in the hour field means "the hours 10, 11 and 12".
- , Used to specify additional values. For example, "MON,WED,FRI" in the day-of-week field means "the days Monday, Wednesday, and Friday".
- / Used to specify increments. For example, "0/15" in the seconds field means "the seconds 0, 15, 30, and 45". And "5/15" in the seconds field means "the seconds 5, 20, 35, and 50". You can also specify '/' after the " character - in this case " is equivalent to having '0' before the '/'. "1/3" in the day-of-month field means "run every 3 days starting on the first day of the month".
C Cron format and examples

- \( * \) ("last") Has a different meaning in each of the two fields in which it is allowed. For example, the value \( L \) in the day-of-month field means "the last day of the month" - day 31 for January, day 28 for February on non-leap years. If used in the day-of-week field by itself, it simply means "7" or "SAT". But if used in the day-of-week field after another value, it means "the last xxx day of the month" - for example "6L" means "the last friday of the month". You can also specify an offset from the last day of the month, such as "L-3" which would mean the third-to-last day of the calendar month. When using the 'L' option, it is important not to specify lists, or ranges of values, as you'll get confusing/unexpected results.

- \( W \) ("weekday") Used to specify the weekday (Monday-Friday) nearest the given day. As an example, if you were to specify "15W" as the value for the day-of-month field, the meaning is: "the nearest weekday to the 15th of the month". So if the 15th is a Saturday, the trigger will run on Friday the 14th. If the 15th is a Sunday, the trigger will run on Monday the 16th. If the 15th is a Tuesday, then it will run on Tuesday the 15th. However if you specify "1W" as the value for day-of-month, and the 1st is a Saturday, the trigger will run on Monday the 3rd, as it will not 'jump' over the boundary of a month's days. The 'W' character can only be specified when the day-of-month is a single day, not a range or list of days. ** The 'L' and 'W' characters can also be combined in the day-of-month field to yield 'LW', which translates to "last weekday of the month".

- \( * \) Used to specify "the nth" XXX day of the month. For example, the value of "6#3" in the day-of-week field means "the third Friday of the month" (day 6 = Friday and "#3" = the 3rd one in the month). Other examples: "2#1" = the first Monday of the month and "4#5" = the fifth Wednesday of the month. Note that if you specify "#5" and there is not 5 of the given day-of-week in the month, then no firing will occur that month. ** The legal characters and the names of months and days of the week are not case sensitive. MON is the same as mon.

C.3 Usage examples

Here are some examples of cron expressions and their effect.

<table>
<thead>
<tr>
<th>Cron expression example</th>
<th>Trigger frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 0 12 * * ?</td>
<td>12 pm (noon) every day</td>
</tr>
<tr>
<td>0 15 10 ? ?</td>
<td>10:15am every day</td>
</tr>
<tr>
<td>0 15 10 * * ?</td>
<td>10:15am every day</td>
</tr>
<tr>
<td>0 15 10 * * ?</td>
<td>10:15am every day</td>
</tr>
<tr>
<td>0 15 10 * * ? 2005</td>
<td>10:15am every day during the year 2005</td>
</tr>
<tr>
<td>0 * 14 * * ?</td>
<td>Every minute starting at 2pm and ending at 2:59pm every day</td>
</tr>
<tr>
<td>0 0/5 14 * * ?</td>
<td>Every 5 minutes starting at 2pm and ending at 2:55pm every day</td>
</tr>
</tbody>
</table>
## Cron format and examples

<table>
<thead>
<tr>
<th>Cron expression example</th>
<th>Trigger frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 0/5 14,18 * * ?</td>
<td>Every 5 minutes starting at 2pm and ending at 2:55pm every day, AND every 5 minutes starting at 6pm and ending at 6:55pm every day</td>
</tr>
<tr>
<td>0 0-5 14 * * ?</td>
<td>Every minute starting at 2pm and ending at 2:05pm every day</td>
</tr>
<tr>
<td>0 10,44 14 ? 3 WED</td>
<td>At 2:10pm and at 2:44pm every Wednesday, in the month of March</td>
</tr>
<tr>
<td>0 15 10 ? * MON-FRI</td>
<td>At 10:15am every day from Monday - Friday</td>
</tr>
<tr>
<td>0 15 10 15 * ?</td>
<td>10:15am on the 15th day of every month</td>
</tr>
<tr>
<td>0 15 10 L * ?</td>
<td>10:15am on the last day of every month</td>
</tr>
<tr>
<td>0 15 10 ? * 6L</td>
<td>10:15am on the last Friday of every month</td>
</tr>
<tr>
<td>0 15 10 ? * 6L 2002-2005</td>
<td>10:15am on the last Friday of every month, during the years 2002-2005</td>
</tr>
<tr>
<td>0 15 10 ? * 6#3</td>
<td>10:15am on the third Friday of every month</td>
</tr>
<tr>
<td>0 0 12 1/5 * ?</td>
<td>12pm (noon) every 5 days every month, starting on the first day of the month</td>
</tr>
<tr>
<td>0 11 11 11 11 *</td>
<td>11:11am on every November 11th</td>
</tr>
</tbody>
</table>
D  Supported characters

To prevent character validation errors, Compose best practice is to only use alphanumeric characters, underscores and hyphens in table and column names. This is because object naming rules are always determined by the database type, of which there may be several in a single Compose project.
Attribute
In the Compose model, an attribute is a logical representation of a physical column in a source database (or Landing Zone) table.

Attributes Domain
A list of all the attributes available in the Compose model. You can add, edit and delete attributes according to your data warehousing needs. The Attributes Domain also shows you which entities each attribute is used in, as a single attribute may be used in several entities.

Change Tables
Change Tables are created in the Landing Zone when the Replicate task is defined as Full Load and Store Changes or Store Changes only. When the Store Changes replication option is enabled in the Replicate task, any changes to the source tables will be replicated to the Change Tables in the Landing Zone. The Change Table name format comprises the original table name appended with a "__ct".

Entity
In the Compose model, an entity is a logical representation of a physical source database/Landing Zone table or view.

ETL Task
In a project, the following ETL tasks can be run: - An ETL task that extracts data from the Landing Zone, performs user-defined transformations on the data, and loads it into the data warehouse tables. - An ETL task that extracts data from the data warehouse, performs user-defined transformations on the data, and loads it into the data mart tables. Depending on the ETL task type and specific settings within Compose, only changes to the existing data will be populated or all of the data (regardless of whether any changes were made to the source).
**Full Load**

A Full Load replication task is a Replicate task that replicates all of the selected source tables to the Landing Zone and populates them with data from the source database. When you duplicate an existing data warehouse ETL, you can set the ETL type to Full Load and Change Tables (i.e. initially extract all the data from the Landing Zone tables and then only the changes), Full Load Only (i.e. extract all the data from the Landing Zone tables) or Change Tables Only (i.e. extract only the changes to the Landing Zone tables).

**History**

Model attributes (and their corresponding data warehouse columns) can either be defined as history Type 1 or history Type 2. When an attribute is defined as history type 1, no history of the data is kept since old data will always be overwritten with new data. When an attribute is defined as history Type 2, a new record is added each time the record is updated. This is especially useful for Slowly Changing Dimensions (SCDs). For example, defining the Address attribute in the Customers table as Type 2 would enable you to retrieve data based on the customer’s location during a certain time period. Attributes defined as history Type 1 will always exist in hub tables whereas attributes defined as history Type 2 will always exist in satellite tables.

**Hub**

A table in the data warehouse containing history Type 1 columns. When a column is defined as history type 1, no history of the data is kept since old data is overwritten with new data.

**Incremental loading**

The activity of loading only new or updated records from the data warehouse into the data mart(s), using Full Load formatted data. The input data may include all of the records (full) or only added and updated records (partial). As opposed to CDC, incremental loading does not indicate whether the change is an UPDATE, INSERT, or DELETE.
Landing Zone
The area in the data warehouse to which the source tables are replicated. This is also the target endpoint in a Replicate task.

Lineage
A visual representation of the data flow of a particular table or column from its source to its current location. Before editing an entity or attribute, you may want to see which other entities/attributes or tables/columns will be impacted by the change. For example, removing the "Discount" attribute from a table will affect the "Total Price". Additionally, a single attribute may have multiple names depending on its location.

Model
The business information model of an enterprise. Usually an ERD (Entity-Relationship Diagram), the model should contain all of the information needed to create the data warehouse. Models can be imported from ERwin or generated automatically by discovering (otherwise known as reverse engineering) the source database or Landing Zone.

Relationship
Similar to a foreign key, a relationship "attribute" is a special type of attribute that points to another entity in the same model.

Satellite
A table in the data warehouse containing history Type 2 columns. When a column is defined as history type 1, a new record is added whenever a record is updated (instead the existing record being overwritten). Satellite tables also contain two additional columns: FD (From Date) and TD (To Date). For old records, these columns show the dates between which a particular record was current (i.e. before a new record rendered it obsolete). The TD column will only
contain a date if the record has been succeeded by a newer record. In Compose, you can set a satellite number (1 and above) for attributes in the model. This is a good way of ensuring that similar attributes (or columns in the data warehouse) appear in the same satellite table. For example, setting the same satellite number for the "Total" and "Discount" attributes ensures that both attributes will be included in the same satellite table.