IBM DataQuant for z/OS and Multiplatforms
Version 2 Release 1

Getting Started with DataQuant

IBM
Contents

Tables ........................................................................... xi

About this information ............................................. xiii
Who should read this information ......................... xiii
Service updates and support information ................. xiii
Highlighting conventions ........................................ xiii
How to read syntax diagrams ................................ xiii
How to look up message explanations ................... xv
How to send your comments ................................ xv

Chapter 1. DataQuant overview .............................. 1
User perspective overview ..................................... 1
Administrator perspective overview ..................... 2

Chapter 2. Using perspectives to set up the User and Administrator application interface 3
Working with views and perspectives ..................... 6
  Opening perspectives ........................................ 6
  Closing perspectives ....................................... 7
  Opening views .............................................. 7
  Positioning views ......................................... 8
  Configuring perspectives ............................... 8
  Resetting perspectives ................................ 8
Setting preferences ............................................. 9
  Setting general preferences ............................ 9
  Setting appearance preferences ..................... 10
  Setting dashboard runtime preferences ............. 10
  Working with global variables ....................... 11
  Setting help preferences ................................ 12
  Setting JDBC preferences ............................ 14
  Setting LOB preferences ................................ 15
  Setting log preferences ................................ 16
  Setting SMTP server preferences ................... 17
  Setting the directory location preference for objects generated by Export actions ..................... 18
  Setting root directory location for dynamart source files ........................................ 19
  Setting third party library preferences ................ 19
  Specifying Visual Designer preferences ................ 20
Exploring the Workspaces view ......................... 21
  Discovering the contents of a workspace .......... 21
  Workspace menu commands ............................ 22
  Managing workspace folders .......................... 23
  Filtering data source objects ......................... 24
  Creating links to workspace objects ................ 24
  Creating links to repository objects ................. 25
  Viewing properties for workspace objects .......... 25
  Refreshing workspace contents ........................ 26
  Moving workspace objects ............................ 26
  Adding content to a workspace ...................... 26
  Creating additional workspaces ..................... 27
Working in the Personal view .................... 28
  Personal view menu commands .................... 28

Chapter 3. Creating queries .................................. 29
Query types ..................................................... 29
Query interfaces ................................................. 30
## Contents

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Creating procedures as flowcharts</td>
</tr>
<tr>
<td>Editing flowchart procedures</td>
</tr>
<tr>
<td>Adding JavaScript conditions to procedures</td>
</tr>
<tr>
<td><strong>Chapter 8. Managing query results.</strong></td>
</tr>
<tr>
<td>Formatting the query results</td>
</tr>
<tr>
<td>Using the Properties view</td>
</tr>
<tr>
<td>Using the Layout Properties dialog box</td>
</tr>
<tr>
<td>Filtering query results</td>
</tr>
<tr>
<td>Working with calculated columns</td>
</tr>
<tr>
<td>Adding calculated columns to a query result set</td>
</tr>
<tr>
<td>Adding custom functions</td>
</tr>
<tr>
<td>Applying categorization to query results</td>
</tr>
<tr>
<td>Adding conformance rules for categorization</td>
</tr>
<tr>
<td>Adding expressions for categorization</td>
</tr>
<tr>
<td>Applying grouping and aggregation to query result columns</td>
</tr>
<tr>
<td>Grouping and aggregation fields</td>
</tr>
<tr>
<td>Exporting query results</td>
</tr>
<tr>
<td>Sending query results in an email</td>
</tr>
<tr>
<td>Exporting query results to a database</td>
</tr>
<tr>
<td>Exporting query results to a file</td>
</tr>
<tr>
<td>Importing query results from CSV and text files</td>
</tr>
<tr>
<td>Specifying separator options for CSV and TXT files</td>
</tr>
<tr>
<td>Specifying the result set structure of CSV and Text files</td>
</tr>
<tr>
<td>Importing data contained in IXF files</td>
</tr>
<tr>
<td>Opening the contents of an IXF file for further processing</td>
</tr>
<tr>
<td>Directly importing an IXF file into a database table</td>
</tr>
<tr>
<td>Importing table contents of a saved file into a database table</td>
</tr>
<tr>
<td>Saving data objects as dynamarts</td>
</tr>
<tr>
<td>Managing source data files in dynamarts</td>
</tr>
<tr>
<td>Working with LOB data in the query results</td>
</tr>
<tr>
<td>Overriding LOB resource limit option</td>
</tr>
<tr>
<td>Viewing and editing the contents of a cell in the query results</td>
</tr>
<tr>
<td>Viewing multiple result sets</td>
</tr>
<tr>
<td>Results menu</td>
</tr>
<tr>
<td>Sorting query results</td>
</tr>
<tr>
<td>Specifying print options for query result sets and quick reports</td>
</tr>
<tr>
<td>Codes to insert specific data in a page header and footer</td>
</tr>
<tr>
<td>Saving data using the regular and fast save methods</td>
</tr>
<tr>
<td><strong>Chapter 9. Working with scheduled tasks</strong></td>
</tr>
<tr>
<td>Adding an execute object action</td>
</tr>
<tr>
<td>Specifying export query operation options</td>
</tr>
<tr>
<td>Specifying export query to mail operation options</td>
</tr>
<tr>
<td>Specifying print query operation options</td>
</tr>
<tr>
<td>Specifying export quick report to mail operation options</td>
</tr>
<tr>
<td>Specifying export quick report operation options</td>
</tr>
<tr>
<td>Specifying export visual report to mail operation options</td>
</tr>
<tr>
<td>Specifying export visual report operation options</td>
</tr>
<tr>
<td>Adding a resource manipulation task</td>
</tr>
<tr>
<td>Specifying copy resource parameters</td>
</tr>
<tr>
<td>Specifying mail settings</td>
</tr>
<tr>
<td>Adding an action group</td>
</tr>
<tr>
<td>Adding actions to an action group</td>
</tr>
<tr>
<td>Removing actions from an action group</td>
</tr>
<tr>
<td>Setting Iterator parameters</td>
</tr>
<tr>
<td>Managing prompts in an action group</td>
</tr>
<tr>
<td>Specifying variables for actions and action groups</td>
</tr>
<tr>
<td>Scheduling local tasks</td>
</tr>
<tr>
<td>Sending HTML tables in email</td>
</tr>
</tbody>
</table>
### Chapter 10. Generating reports

- Creating a new report ........................................... 159
- Selecting a classic report's data source object ................. 159
- Listing data source objects ...................................... 160
- Exporting reports to a file .................................... 161
- Creating classic report forms .................................. 162
  - Designing a classic report form ............................ 163
  - Using form variables ........................................ 163
  - Using usage codes .......................................... 175
- Saving classic reports .......................................... 176
  - Saving a form with an associated data source object ...... 179
  - Saving a form without an associated data source object .. 179
- Working with quick reports .................................. 180
  - Creating quick reports .................................... 180
  - Editing quick reports ...................................... 197
  - Modifying your quick report design ......................... 199
- Form menu ..................................................... 200

### Chapter 11. Working with forecasts

- Creating forecasts ............................................. 201
- Specifying forecast data source options ...................... 203
- Specifying forecast date parameters ........................ 203
- Specifying forecast grouping hierarchy options .......... 204
- Specifying forecast strategy options ........................ 205
- Specifying forecasting models options ....................... 205
  - Setting simple moving average parameters ............... 206
  - Setting weighted moving average parameters ............ 207
  - Setting single exponential smoothing parameters ...... 207
  - Setting double exponential smoothing parameters ..... 207
  - Setting Holt-Winters method parameters ................. 208
  - Setting multiplicative decomposition parameters ...... 208
  - Setting curve fitting parameters .......................... 209
  - Setting polynomial regression parameters .............. 209
  - Setting neural network parameters ........................ 210
- Monitoring forecast model performance ..................... 210
- Observing trends .............................................. 211
- Observing seasonality ......................................... 212
- Observing cyclicity .......................................... 212
- Editing chart properties ..................................... 213
- Editing grid properties ..................................... 213
- Exporting forecasts ............................................ 214
- Specifying print options for a forecast ..................... 214

### Chapter 12. Working with charts

- Creating charts .................................................. 217
  - Specifying chart type options .............................. 218
  - Specifying data series options ............................. 218
  - Specifying format options .................................. 219
- Creating charts from a query results grid .................. 226
- Working with charts in the Display editor ................. 227
  - Creating charts in the Display editor ..................... 227
  - Managing display modes .................................... 228
  - Editing query result set columns ......................... 228
  - Editing display mode layout structures ................... 229
- Editing charts .................................................... 230
- Exporting charts ................................................ 230
- Specifying print options for charts ......................... 231

### Chapter 13. Working with visual projects

- Visual project templates ....................................... 234
### Tables

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>User perspective: Views</td>
</tr>
<tr>
<td>2.</td>
<td>Administrator perspective: Views</td>
</tr>
<tr>
<td>4.</td>
<td>Text Analytics: Views</td>
</tr>
<tr>
<td>5.</td>
<td>Columns and available actions</td>
</tr>
<tr>
<td>6.</td>
<td>Prompt types</td>
</tr>
<tr>
<td>7.</td>
<td>Original result set data</td>
</tr>
<tr>
<td>8.</td>
<td>Normalized query results</td>
</tr>
<tr>
<td>9.</td>
<td>Alternate normalized query results</td>
</tr>
<tr>
<td>10.</td>
<td>Crosstabbed query results for furniture sales</td>
</tr>
<tr>
<td>11.</td>
<td>Normalized query results for furniture sales</td>
</tr>
<tr>
<td>12.</td>
<td>Original data</td>
</tr>
<tr>
<td>13.</td>
<td>Conditionally grouped data</td>
</tr>
<tr>
<td>14.</td>
<td>Text options</td>
</tr>
<tr>
<td>15.</td>
<td>Query result set and quick report print options</td>
</tr>
<tr>
<td>16.</td>
<td>Codes to insert specific data</td>
</tr>
<tr>
<td>17.</td>
<td>Main section elements for a classic report</td>
</tr>
<tr>
<td>18.</td>
<td>Breaks section elements for a classic report</td>
</tr>
<tr>
<td>19.</td>
<td>HTML variables</td>
</tr>
<tr>
<td>20.</td>
<td>Calculations option elements for a classic report</td>
</tr>
<tr>
<td>21.</td>
<td>Columns option elements for a classic report</td>
</tr>
<tr>
<td>22.</td>
<td>Conditions option elements for a classic report</td>
</tr>
<tr>
<td>23.</td>
<td>Detail section elements for a classic report</td>
</tr>
<tr>
<td>24.</td>
<td>Options section elements for a classic report</td>
</tr>
<tr>
<td>25.</td>
<td>Final section elements for a classic report</td>
</tr>
<tr>
<td>26.</td>
<td>Page section elements for a classic report</td>
</tr>
<tr>
<td>27.</td>
<td>Buttons and their indexes</td>
</tr>
<tr>
<td>28.</td>
<td>Toolbar options</td>
</tr>
<tr>
<td>29.</td>
<td>Command line parameters</td>
</tr>
</tbody>
</table>
About this information

This information describes how to use the DataQuant for Workstation and DataQuant for WebSphere® applications.\(^1\)

Always check the DB2® and IMS™ Tools Library page for the most current version of this publication:

www.ibm.com/software/data/db2imstools/db2tools-library.html

Who should read this information

This information is intended for all DataQuant for Workstation and Dataquant for WebSphere users.

Service updates and support information

To find service updates and support information, including software fix packs, PTFs, Frequently Asked Questions (FAQs), technical notes, troubleshooting information, and downloads, refer to the following web page:

http://www.ibm.com/support/entry/portal/Overview/Software/Information_Management/DB2_Tools_for_z~OS

Highlighting conventions

This information uses the following highlighting conventions:

- **Boldface** type indicates commands or user interface controls such as names of fields, folders, icons, or menu choices.
- **Monospace** type indicates examples of text that you enter exactly as shown.
- **Italic** indicates the titles of other publications or emphasis on significant terms. It is also used to indicate variables that you should replace with a value.

How to read syntax diagrams

The following rules apply to the syntax diagrams that are used in this information:

- Read the syntax diagrams from left to right, from top to bottom, following the path of the line. The following conventions are used:
  - The >>>-- symbol indicates the beginning of a syntax diagram.
  - The -->> symbol indicates that the syntax diagram is continued on the next line.
  - The >>-- symbol indicates that a syntax diagram is continued from the previous line.
  - The -->>< symbol indicates the end of a syntax diagram.
- Required items appear on the horizontal line (the main path).

---

\(^1\) Throughout this information, the IBM® DataQuant client for WebSphere Application Server is referred to as DataQuant for WebSphere and the IBM DataQuant client for the workstation environment is referred to as DataQuant for Workstation.
• Optional items appear below the main path.

```
►►—required_item—◄◄
    ^optional_item^
```

If an optional item appears above the main path, that item has no effect on the execution of the syntax element and is used only for readability.

```
►►—required_item—◄◄
    ^optional_item^
```

• If you can choose from two or more items, they appear vertically, in a stack.

If you must choose one of the items, one item of the stack appears on the main path.

```
►►—required_item—◄◄
    —required_choice1—
    —required_choice2—
```

If choosing one of the items is optional, the entire stack appears below the main path.

```
►►—required_item—◄◄
    ^optional_choice1^
    ^optional_choice2^
```

If one of the items is the default, it appears above the main path, and the remaining choices are shown below.

```
►►—required_item—◄◄
    —default_choice—
    ^optional_choice^
    ^optional_choice^
```

• An arrow returning to the left, above the main line, indicates an item that can be repeated.

```
►►—required_item—◄◄
    ▼repeatable_item
```

If the repeat arrow contains a comma, you must separate repeated items with a comma.

```
►►—required_item—◄◄
    ▼repeatable_item
```

A repeat arrow above a stack indicates that you can repeat the items in the stack.
How to look up message explanations

You can use any of the following methods to search for messages and codes.

Searching an information center

In the search box that is in the upper left toolbar of any Eclipse help system, such as the IBM Information Management Software for z/OS Solutions Information Center, enter the number of the message that you want to locate. For example, you can enter DFS1065A in the search field.

Use the following tips to help you improve your message searches:

- You can search for information about codes by entering the code; for example, enter -327.
- Enter the complete or partial message number. You can use wildcards in the message number to broaden your search; use * to represent multiple characters and use ? to represent any single character. For example:
  - The search string DFS20?I returns any messages that begin with the string DFS20, followed by any single character, followed by I.
  - The search string DFS20??I returns any messages that begin with the string DFS20, followed by any two characters, followed by I.
  - The search string DFS20+I returns any messages that begin with the string DFS20, followed by any number and type of characters, followed by I.

The information center contains the latest message information for all of the information management products that are included in the information center.

Using a web search engine

You can use any of the popular search engines that are available on the web to search for message explanations. When you type the specific message number or code into the search engine, you are presented with links to the message information in IBM information centers.

How to send your comments

Your feedback is important in helping to provide the most accurate and high-quality information. If you have any comments about this book or any other documentation, use either of the following options:

- Use the online reader comment form, which is located at:
- Send your comments by e-mail to comments@us.ibm.com. Be sure to include the name of the book, the part number of the book, the version of your product,
and, if applicable, the specific location of the text you are commenting on (for example, a page number or table number).
Chapter 1. DataQuant overview

DataQuant is an all-purpose, multi-platform, Business Intelligence tool.

DataQuant is an Eclipse-based Business Intelligence tool that transforms business data into a visual information platform with visual data on-demand. Using the DataQuant feature set, users can create intuitive dashboards, queries, reports, and procedures that can be easily distributed across an entire enterprise.

- Delivers critical enterprise-wide business information across multiple user and database platforms.
- Supports DB2 V9 for z/OS support for SQL stored procedures.
- Supports DB2 V10 for z/OS functionality, including DB2 V10 data types.
- Supports dynamic and static SQL, massive result sets, and stored procedure results.
- Supports simultaneously drawing data from DB2, Informix, Oracle, SQL Server and most other JDBC-compliant data sources, as well as multi-dimensional data sources that support XXMLA connectivity.
- Reports, dashboards, and queries can be optionally embedded within custom or 3rd party web applications, portals or workstation applications.
- Supports multi-dimensional queries, dashboards, reports, and layouts, as well as enhanced interoperability with existing BI infrastructure.
- DataQuant for Workstation is an Eclipse-based, cross-platform, rich workstation application that can be run on Microsoft Windows, Linux, and Macintosh.
- DataQuant for WebSphere delivers broad functionality to enterprise users through ordinary Web browsers.

User perspective overview

The User perspective of the application interface provides the views, menus and wizards that enable you to perform an extensive range of query and reporting functions.

About this task

To get started using DataQuant, you should become familiar with the User perspective, the user preferences that you can set, and how you can work with the data sources and objects that are available to you in the Workspaces view. At any time, you can begin querying data sources, creating objects, formatting query results, or creating reports and dashboards.

Note: The application interface configuration information consists of the JDBC driver and repository connection settings that are used to connect to shared

2. Eclipse is an open source community whose projects are focused on providing an extensible development platform and application frameworks for building software. Eclipse provides extensible tools and frameworks that span the software development lifecycle, including support for modeling, language development environments for Java, C/C++, PHP and others, testing and performance, business intelligence, rich client applications and embedded development. A large, vibrant ecosystem of major technology vendors, innovative start-ups, universities and research institutions and individuals extend, complement and support the Eclipse Platform. The Eclipse Foundation is a not-for-profit, member supported corporation that hosts the Eclipse projects. Full details of Eclipse and the Eclipse Foundation are available at [www.eclipse.org](http://www.eclipse.org)
repositories and data sources. Advanced users can make changes to the configuration information from the User perspective. Most users will never change the configuration information.

Administrator perspective overview

The Administrator perspective of the application interface provides the views, menus, and wizards that enable you to create, manage, and maintain repositories.

A DataQuant administrator is responsible for setting up your application interface's configuration. As a result, the application interface that you are using is installed preconfigured with access to one or more repository workspaces. The data sources and objects that you can access from the workspace are displayed in the Workspaces view.
Chapter 2. Using perspectives to set up the User and Administrator application interface

Use Perspectives, Views and Editors to set up and work in the user and administrator environments.

**Perspectives**

DataQuant utilizes and employs the Eclipse-based concept of Perspectives, Views, and Editors.

The sections that follow explain Perspectives, Views, and Editors as used by DataQuant. For information about the Eclipse concept of Perspectives, Views and Editors consult the [Eclipse documentation](#).

A Perspective is a way to organize and view the files and artifacts used with DataQuant.

The Perspectives that come with DataQuant are configured with set of default views, that are docked to specific areas in the application interface.

Administrators and users can select the Perspectives, Views, and Editors that suit their roles. For example, if your primary role is to design and develop visual dashboards, you will likely perform much of your work in the Visual Designer perspective.

Additionally, administrators and users can customize the interface so that whenever they launch DataQuant, it opens to a specific perspective and set of views.

To see the Perspectives available for use with DataQuant, perform **Window > Open Perspective > Other** from the menu bar.

DataQuant for Workstation and DataQuant for WebSphere provide the following perspectives:

- **Administrator** which provides the views, menus, and wizards that enable a user with administrative privileges to create and maintain repositories. Not all instances of DataQuant are installed with the Administrator perspective as a selected feature.
- **User** which provides the views, menus, and wizards that you will use to query data sources and work with query results.
- **QMF** which provides the views, menus, and wizards that are most appropriate for working with QMF™ catalogs.
- **Visual Designer** which provides the views, menus, and wizards that you will use to create visual reports and dashboards.
- **Text Analytics** which provides the interface from which you create and work with Text Analytics projects.
**Views**

A View is typically used to navigate a hierarchy of information, open an editor, or display properties for the active editor.

DataQuant views support editors and provide alternative presentations as well as ways to navigate the information in your window. In Eclipse, the layout of views within a page is controlled by the active perspective. This means that each of the perspectives (User, Visual Designer, QMF, and Administrator) have a designated or predetermined set of Views. Each view displays as a pane in the main workstation window. Each perspective of DataQuant initially displays the views that are most commonly used to perform specific functions.

*Table 1. User perspective: Views*

<table>
<thead>
<tr>
<th>View Name</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analyzer</td>
<td>The Analyzer view displays the weight count of each object lying on a scene.</td>
</tr>
<tr>
<td>Bookmarks</td>
<td>The Bookmarks view manages bookmarks for drill-down paths.</td>
</tr>
<tr>
<td>Cheat Sheets</td>
<td>The Cheat Sheets view displays a series of tutorials that outline the steps required to implement several basic user and administrator functions.</td>
</tr>
<tr>
<td>Data Source Connections</td>
<td>The Data Source Connections view displays a list of all connections that are currently active or were created to a data source.</td>
</tr>
<tr>
<td>Help</td>
<td>The Help view launches the Help for the application.</td>
</tr>
<tr>
<td>Outline</td>
<td>The Outline view displays the structural elements of the object that is currently open in the editor area.</td>
</tr>
<tr>
<td>Output</td>
<td>The Output view displays informational and error messages generated while running the application interface.</td>
</tr>
<tr>
<td>Personal</td>
<td>The Personal view maintains and displays objects in the Recently Used, Favorites, and Startup folders.</td>
</tr>
<tr>
<td>Profiler</td>
<td>The Profiler view displays the data on query running, the amount of created objects, and the time spent on creating them and calculating expressions on the visual project.</td>
</tr>
<tr>
<td>Progress</td>
<td>The Progress view displays all progress, percentage of work completed, and cancellation messages.</td>
</tr>
<tr>
<td>Properties</td>
<td>The Properties view displays the properties of the object that is selected or active in the editor area or another view.</td>
</tr>
<tr>
<td>Query Profiler</td>
<td>The Query Profiler view displays the time spent on running a query and fetching its results.</td>
</tr>
<tr>
<td>Repository Connections</td>
<td>The Repository Connections view displays the list of repository connections that you have available.</td>
</tr>
<tr>
<td>Repository Explorer</td>
<td>The Repository Explorer view displays the contents of the repositories to which you have connected. It also displays information about the references one object has on other objects.</td>
</tr>
<tr>
<td>Results Navigator</td>
<td>The Results Navigator view allows switching between results of a query or drill-down path.</td>
</tr>
<tr>
<td>REXX console</td>
<td>The REXX console view supports work with REXX procedures.</td>
</tr>
<tr>
<td>Search</td>
<td>The Search view displays the results of a search.</td>
</tr>
</tbody>
</table>
Table 1. User perspective: Views (continued)

<table>
<thead>
<tr>
<th>View Name</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>Workspaces</td>
<td>The Workspaces view lists the workspaces that are available to your user ID. All of the data sources that you can access are contained in the workspaces that are presented. From the Workspaces view you can perform the majority of query and reporting functions.</td>
</tr>
</tbody>
</table>

Table 2. Administrator perspective: Views

<table>
<thead>
<tr>
<th>View</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>Repositories</td>
<td>The Repositories view displays the repositories that you created. This view is only available if you have the Administrator component.</td>
</tr>
<tr>
<td>Users and Groups</td>
<td>The Users and Groups view displays the users and groups created for your internal security provider.</td>
</tr>
</tbody>
</table>

Table 3. Visual Designer perspective: Views

<table>
<thead>
<tr>
<th>View</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>Events</td>
<td>The Events view displays navigation elements that you can apply to layout objects within a visual project.</td>
</tr>
<tr>
<td>Expressions</td>
<td>The Expressions view is used for debugging visual dashboards.</td>
</tr>
<tr>
<td>Layers</td>
<td>The Layers view displays the groups of layers.</td>
</tr>
<tr>
<td>Palette</td>
<td>The Palette view displays the objects that you can insert in a visual project.</td>
</tr>
<tr>
<td>Project Explorer</td>
<td>The Project Explorer view displays each element in a visual report and dashboard.</td>
</tr>
<tr>
<td>Properties</td>
<td>The Properties view displays the properties of the object that is selected or active in the editor or another view.</td>
</tr>
<tr>
<td>DataQuant Calculator</td>
<td>The DataQuant Calculator is used for calculating expressions in visual projects.</td>
</tr>
</tbody>
</table>

Table 4. Text Analytics: Views

<table>
<thead>
<tr>
<th>View Name</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>Entities Explorer</td>
<td>The Entities Explorer view displays the tree structure of the entities which are specified in the knowledge base.</td>
</tr>
<tr>
<td>Source Explorer</td>
<td>The Source Explorer view displays the tree structure of the files, belonging to the projects.</td>
</tr>
<tr>
<td>Text Analytics Explorer</td>
<td>The Text Analytics Explorer view displays the list of projects that can be analyzed.</td>
</tr>
<tr>
<td>Word Map</td>
<td>The Word Map view displays the text analysis results in the form of a word cloud.</td>
</tr>
</tbody>
</table>

Editors

Depending on the type of file that is being edited, the appropriate editor is displayed in the editor area.

Primary functional differences between views and editors:
• There is generally only one instance of a given view per workbench page, but there can be several instances of the same type of editor.
• Editors can appear in only one region of the page, whereas views can be moved to any part of the page and minimized as fast views.
• Editors can be in a dirty state, meaning that their contents are unsaved and will be lost if the editor is closed without saving.
• Views have a local toolbar, whereas editors contribute buttons to the global toolbar.
• Editors can be associated with a file name or an extension, and this association can be changed by users.

Each of the DataQuant perspectives includes a single editor area. By default, the editor area opens in the center of the workstation window.

There are many editors in DataQuant that will open in the editor area based on what type of object you open or what type of function you perform. You will use many of these editors when you are writing queries, formatting query results, creating reports or creating visual dashboards.

Working with views and perspectives
The DataQuant query environment includes views and perspectives that can be customized to fit your needs.

The following sections describe how to:
• Open and close perspectives and views
• Position and dock views in the application interface
• Customize the application interface to open to a perspective and view that suits your role and responsibility.

Opening perspectives
You can open any perspective that is not currently open in the workstation window.

About this task
A perspective will stay open until you close that perspective. Each time DataQuant is started, all perspectives that were open at the time of the previous close will open.

Initially, the User perspective opens when the DataQuant application is started. If the User perspective is closed you can open it, or any perspective that has been closed, in the following way:

Procedure
1. Click the Open Perspective toolbar button. You can also select Window > Open Perspective. A menu that lists the perspectives that you have opened in the current DataQuant session is displayed. Select the perspective that you want to open.
2. To see a complete list of available perspectives, select Other from the menu. The Open Perspective window opens. Select the perspective that you want to open. Click OK.
3. The perspective that you have selected opens. A tab is added to the title bar of the window. The tab displays the name of the perspective and its associated button. Clicking the button for the perspective allows you to quickly switch to a perspective from other perspectives in the same window.

4. By default, a perspective will open in the same window. You can open a perspective in a new window by selecting Window > Open New Window. The active perspective opens in a new window.

**Closing perspectives**

You can close each perspective that is active in the workstation window.

**About this task**

If you close a perspective, it will stay closed until you re-open that perspective. Each time a DataQuant application interface is started, all perspectives that were open at the time of the previous close will open, all perspectives that were closed will stay closed.

To close a perspective:

**Procedure**

1. Select Window > Close Perspective. The active perspective closes.
2. To close all perspectives, select Window > Close All Perspectives. All of the perspectives that are open in the workstation window are closed. The workstation window appears empty.

**Opening views**

You can change the layout of a perspective by opening and closing the views that are displayed in the workstation window.

**About this task**

To open a view:

**Procedure**

1. Select Window > Show View.
2. From the list of views, select the view that you would like to open. Choose Other to see additional views. The Show View window opens. Select the view that you want to open.
3. Click OK. The selected view opens in the perspective that is active in the workstation window. It will be opened automatically with the perspective in subsequent DataQuant sessions.

**What to do next**

**Note:** Views also have their own menus. To open the menu for a view, click the icon at the left end of the view's title bar. Some views also have their own toolbars. The actions represented by buttons on view toolbars only affect the items within that view.
Positioning views

If you are using DataQuant for Workstation, you can change the layout of a perspective by opening and closing views and by docking them in different positions in the workstation window.

About this task

When opened, a view might appear by itself, or stacked with other views in a tabbed notebook. You can change the location of a view in one of the following ways:

- Drag the view by its title bar to the new location and release the left mouse button.
- Right-click the title bar of the view that you want to move. Select Move > View or Move > Tab Group from the pop-up menu.

Configuring perspectives

In addition to configuring the layout of a perspective in the application interface, you can customize the options that are available from the Window menu; as well as the buttons and menu options that are available from the toolbar and the menu bar.

About this task

The aspects of a perspective that you can control:

- The buttons available on the toolbar.
- The options available in the menus of the menu bar.
- The command groups of the perspective.
- The shortcuts that are available in the perspective.

To configure a perspective:

Procedure

1. Switch to the perspective that you want to configure.
3. Select the tab that contains the options that you want to customize.
4. Expand the item that you want to customize.
5. Use the check boxes to select the elements that you want to see on drop-down menus in the selected perspectives. Items that you do not select will still be accessible by clicking the Other menu option.
6. Click OK.

Resetting perspectives

If you are using DataQuant for Workstation, you can restore a perspective to its original layout.

About this task

To restore a perspective to its original layout:
Procedure
1. Select Window > Reset Perspective. An informational window opens asking whether you want to reset the current perspective to its default values.
2. Click OK. The current perspective is reset to its default values.

Setting preferences

The Preferences dialog is used to set user preferences that will apply to specific DataQuant objects and functions.

The Preferences window consists of two panes. The left pane displays the list of preferences groups and the right pane displays the page for the selected group. The following groups of preferences are displayed in the Preferences window.

Setting general preferences

You use the General page of the Preferences window to specify the language, orientation direction, locale settings, and object default actions.

About this task

To specify the general user preferences for the current DataQuant application interface session:

Procedure
1. Select View > Preferences. The Preferences window opens. Select General from the tree.
2. Conditional: If available as an option on the Preferences window, select from the Language drop-down list of supported languages the language that will be used for the current application interface session. Select the value Auto to use the local language of the current machine.
3. Select from the Direction drop-down list the orientation that will be used to display information in the current DataQuant application interface session. You can select Left-to-Right or Right-to-Left. You can select Auto to use the default orientation of the current machine.
4. Select from the Locale setting for numbers, currencies, times, and dates list the language that will be used to display numbers, currencies, times and dates in the current DataQuant application interface session. Select the value Auto to use the local language of the current machine.
5. Click one of the Default Action for Objects radio buttons to specify the action that DataQuant will perform when you double-click an object that is included in an explorer tree. Select Run object to run the object that is selected. Select Display object to display the SQL content of the object.
6. In the Default Tab for the Query Editor area, you can specify the tab on which the Query Editor is opened by default.
7. Click the Line width ellipsis (…) or enter the value of the maximum line width. The available values for line width are from 10 to 1000. The default value is 79. This value is used in the Query menu item Reformat Text for SQL text.
8. Click Apply to save your preferences choices.
9. Click Restore Defaults to restore the default preference values.
10. Click OK to close the Preferences window.
Setting appearance preferences

You use the Appearance page of the Preferences window to specify default fonts, color scheme, and strings for displaying and entering special values. On the page, you can also select saving custom window size.

About this task

To set the appearance preferences:

Procedure

1. Select View > Preferences. The Preferences window opens. Select Appearance from the tree.
2. In the Default Fonts area, define the font type for headings and columns of query results, classic reports.
3. Specify a string in the Display null values as field that DataQuant will use to display null values that are retrieved from a database data source.
4. Specify a string in the Enter null values as field that you will use when you enter null values in the query results or in database tables using the table editor.
5. Specify a string in the Enter default values as field that you will use when you enter default values in the query results or in database tables using the table editor.
6. Select from the Color scheme in text editors drop-down list what type of color scheme will be supported in the text editors. In most instances you will choose DataQuant to specify that the default color schemes used by the text editors are acceptable. If you will be working in high contrast mode, you will need to select system default.
7. If you customize the size of the windows in the application and want to save these changes, in the Window Size area, select Save custom window size check box. To revert your changes, clear the check box.
8. Click Apply to save your preferences choices.
9. Click Restore Defaults to restore the default preference values.
10. Click OK to close the Preferences dialog box.

Setting dashboard runtime preferences

You use the Dashboard Runtime page of the Preferences dialog to specify the Google Map API key.

About this task

To specify the dashboard runtime user preferences for the current DataQuant application interface session:

Procedure

1. Select View > Preferences. The Preferences window opens. Select Dashboard Runtime from the tree.
2. Set values for the Dashboard Runtime: You can obtain the Google Map APIs from the Google Code web site.
   - In the Google Maps JavaScript API key field, specify the Google Maps JavaScript API key to enable the usage of the Google Map layout object in visual dashboards in the HTML5 rendering mode.
You can obtain the Google Maps JavaScript APIs from the Google Developers web site.

- Set the **Display null values as** to specify how null values are displayed in the dashboards.

3. Click **Apply** to save your preferences choices.
4. Click **Restore Defaults** to restore the default preference values.
5. Click **OK** to close the Preferences window.

### Working with global variables

Global variables are variables that stay active while the current session of DataQuant is active. This is in contrast to substitution variables that are active only during the execution of an object (query, form, procedure).

For objects that use substitution variables, values are entered via a window when the object is run. After the object runs, the variable and the value that you used no longer exists. For objects that use global variables, the value currently defined for the global variable is used.

There are two types of global variables:

- **System global variables** are pre-loaded with the DataQuant application. That means their values are re-initialized each time you open the DataQuant application. System global variables begin with the letters DSQQW, DSQAO, DSQEC, DSQDC, and DSQCP. You cannot add or delete system global variables. However, you can edit the default values for the system global variables.

- **User global variables** are defined by the user. User global variables can be specified with any unique name that does not begin with the letters DSQQW, DSQAO, DSQEC, DSQDC, and DSQCP. User global variables values are not saved when you terminate the DataQuant session.

Using the Global Variables page of the Preferences window you can view, add, delete and edit your user global variables. You can also edit the values of the system global variables.

### Specifying user global variables

You use the Global Variables page of the Preferences window to view, add, delete and edit your user global variables.

### About this task

To view, add, edit and delete user defined global variables:

**Procedure**


2. The system and user global variables that are defined for your session are listed in the **Variables** list box. The name of the global variable is listed in the **Name** field. The current value for the global variable is listed in the **Value** field.

3. To add a new user defined global variable, click the **Add** button. The New variable window opens. Type the name of the new user global variable in the **Variable name** field. You can type any combination of unique characters that have not already been used. You can not begin a user global variable with the system global variable prefixes of DSQQW, DSQAO, DSQEC, DSQDC, and DSQCP. Click **OK**. The user global variable is added to the list and will be
available for use during the current DataQuant session. To specify the initial value of the variable, edit the Value cell as described in the following step.

4. To edit the value of a user global variable, click in the Value field for the global variable that you want to change. Type the new value for the global variable in the Value field.

5. To delete a user global variable, select an existing user global variable from the Variables list and click the Remove button. The user global variable is deleted.

6. Click Apply to save your preferences choices.

7. Click Restore Defaults to restore the default preference values.

8. Click OK to close the Preferences dialog.

**Editing system global variable values**

You use the Global Variables page of the Preferences dialog to edit any system global variable values.

**About this task**

To edit system global variable values:

**Procedure**


2. The system global variables are listed in the Variables list by name. System global variable start with the prefix of DSQQW, DSQAO, DSQEC, DSQDC, or DSQCP.

3. The current system global variable values are listed in the Value field. To edit the value of a global variable, click in the Value field for the system global variable that you want to change. Type the new value for the system global variable in the Value field.

4. Click Restore Defaults to restore the default values for all the system global variables.

5. Click Apply to save your preferences choices.

6. Click OK to close the Preferences window.

**Setting help preferences**

If you are using DataQuant for Workstation, you use the Help page of the Preferences window to specify the default values for how help information is displayed and to change the interface and port specification for the internal server that the help system uses.

**About this task**

To change the web browser adapter setting:

**Procedure**

1. Select View > Preferences. The Preferences window opens. Select Help from the tree.

2. The help system can display information in either the Help view or in a browser. If the help is displayed in a browser, the browser can be an embedded browser or a full external browser. If the embedded browser is supported on
your system, help will use it by default to display help. If you prefer to always use a full external browser, select **Use external browser** from the drop-down list.

3. By default, context-sensitive help for workstation windows is displayed in the **Help** view. If you prefer to display workstation window context-sensitive help in infopops, select **Open window context help in an infopop** from the drop-down list.

4. By default, context-sensitive help for dialogs is displayed in the dialog's tray, which is similar to the **Help** view. If you prefer to display dialog context-sensitive help in infopops, select **Open dialog context help in an infopop** from the drop-down list.

5. Click **Apply** to save your preferences choices.

6. Click **Restore Defaults** to restore the default preference values.

7. Click **OK** to close the Preferences window.

**Including help content from a remote server**

The help system includes an internal server that serves the help contents to the browser. You use this preference page to change the interface and port that the internal server uses.

**About this task**

You should only change these settings if you experience problems and cannot view help with the default preferences.

To change the interface and port that the help server uses:

**Procedure**

1. Select **View > Preferences**. The Preferences dialog opens. Expand the **Help** branch by clicking the plus sign. Select **Content** from the tree.

2. Select the **Include help content from a remote infocenter** check box. The Remote Infocenter window becomes available.

3. Click **Add**. The Add new infocenter window opens.

4. Specify the name of a local IP interface to be used by the server in the **Name** field.

5. Specify the host of a local IP interface to be used by the server in the **Host** field.

6. Specify the URL of a local IP interface to be used by the server in the **Path** field.

7. Select **Use port** and specify the number of a specific port the server will listen to.

8. Click **OK**. The Add new infocenter window closes and control returns to the Preferences window. The new infocenter is displayed in the Remote Infocenters window.

9. To edit an infocenter, click **Edit**. The Edit [infocenter name] infocenter window opens.

10. To delete and infocenter, click **Delete**. The selected infocenter is removed from the list.

11. To view the properties of a specific infocenter, click **View Properties**. The Properties for [infocenter name] window opens with the name, host, path, port, and URL displayed.
12. To test an infocenter’s connection to the local IP interface, click **Test Connection**. The Test Connection window opens displaying the test results.

13. To disable an infocenter without deleting it, click **Disable**. The infocenter is now disabled, and its status in the **Enabled** column is changed to disabled.

14. Click **Apply** to save your preferences choices.

15. Click **Restore Defaults** to restore the default preference values.

16. Click **OK** to close the Preferences window.

### Setting JDBC preferences

You can use the Preferences window to set up the JDBC driver information.

#### About this task

DataQuant administrators are responsible for distributing the JDBC driver information that users will need in order to access any data source. In most cases, administrators distribute this information by deploying a preconfigured JDBC library settings file when the application interface that you are using is installed.

Advanced users can define or change JDBC driver location information using the **JDBC Libraries** page of the Preferences window to specify the JDBC driver information for each type of database (such as DB2, Informix, Oracle) that will be accessed. The JDBC driver locations must be accessible to DataQuant. DataQuant will retain the JDBC driver information. You would only repeat this specification process to add new drivers or make changes to the properties of an existing driver.

To define JDBC driver information:

#### Procedure

1. Select **View > Preferences** to open the Preferences window. Select **JDBC Libraries**. The JDBC Libraries page opens. All of the JDBC drivers and their associated JAR files that you have already set up are listed. They are organized by library.

2. To add new JDBC driver information for a type of database that is not listed, select **Add Driver**. The Add New Driver Library window opens.

3. The JDBC driver information for each type of database is organized by libraries. Enter a name for the library in the **Enter new library name** field. It is recommended that the name that you specify describes the JDBC information that will be included in the library. For example, if you are adding JDBC driver information for accessing DB2 databases, you might call the library DB2. However, this is a descriptive field and can include any text.

4. Specify the actual name of the driver class that will be used in the **Driver class name** field. This can be found in your JDBC driver documentation. For example: com.ibm.db2.jcc.DB2Driver.

5. Optional: Specify a generic example of a correctly formatted URL that could be used to connect to the database in the **URL templates** field. For example, if you are adding JDBC driver information for accessing DB2 databases, you might specify the following JDBC URL template: jdbc:db2://host:50000/database. The generic information as specified in the template is presented when you are adding data sources to the repository and when you are creating repository connections. You will replace the generic information with the specific database information.

6. If you are using DataQuant for Workstation, add the location information for the JDBC driver files to the library. Click **Add JARS**. The Add JARS to
[libraryname] window opens. Search for and select the JDBC driver files that you want to add. Click Open. The location of the JDBC driver files is saved in the JDBC library that you have selected.

7. Click Apply. The JDBC library information that you have specified is saved.

8. You can make the following changes to the JDBC driver information:
   • To edit the JDBC driver library information, select an existing JDBC driver library from the list and click Edit. The Edit Driver Library window opens where you can make changes to the Library name, Driver class name, and JDBC URL template fields.
   • If you are using DataQuant for Workstation, to edit the name and location of the files that contain the JDBC drivers, select a file from the list of files that have been added to the JDBC library, and click Edit. The Edit JAR URL window opens. Specify the name and location in the Edit JAR location field.
   • To remove a JDBC driver library, select an existing JDBC driver library from the list and click Remove. The library, including all the JAR files that it contains, is removed.
   • If you are using DataQuant for Workstation, to remove a JAR file from a JDBC driver library, select an existing file from the list and click Remove. The file is removed.

9. Click OK. The JDBC library information is saved. The Preferences window closes. You must repeat this process for each type of JDBC driver that will be used to access various database data sources such as Informix or SQL Server.

Setting LOB preferences

You use the LOBs page of the Preferences dialog to specify options that will control retrieving and saving LOB data.

About this task

LOB data (Large Object data) is large text or binary objects in a database. LOBs can be large strings of text, images, and so on. If a query result set contains a large amount of LOB data, processing resources could be severely strained. The LOB preferences page contains a number of LOB data retrieval restrictions that you can use to manage repository resources. To specify LOB data preferences:

Procedure

1. Select View > Preferences. The Preferences window opens. Select LOBs from the tree.

2. Check Override LOB options if possible to grant yourself the ability to override the LOB options that have been specified by your DataQuant administrator for your resource limits group. Your administrator must have given your resource limits group permission to override the LOB options. If you do not have permission to override LOB options, this check box is not available.

3. Use the LOB Retrieval Option radio buttons to specify how large object data (LOB) will be retrieved. There are four choices:
   • Select Disable LOB columns to disable LOB data retrieval. By selecting this option, you will not be able to query any table that contains LOB data.
   • Select Disable LOB data retrieval to disable LOB data retrieval but allow access to the other columns in the tables that contain LOB data. By selecting this option, you can query tables that contain LOB data and result data for all columns except those that contain LOBs will be returned.
• Select **Retrieve LOB data on demand** to specify that you want to retrieve all LOB data from a table but only display selected columns of the LOB data in the query results.

By selecting this option, you can query tables that contain LOB data and your result data will be returned for all columns that contain LOB objects. To conserve resources, however, the retrieved LOB data is stored in a file. Pointers (<LOB LOCATOR>) to the LOB data objects are displayed in the query results. You click the pointer to view the LOB data.

You can also pull the data from the database by double-clicking on the pointer.

• Select **Retrieve LOB data automatically** to specify that you want to retrieve all LOB data from a table and immediately display the retrieved LOB data in the query results.

This option pulls ALL of the LOB data for ALL of the LOB columns from the database to the local computer. The actual LOB data is not displayed in grids and reports. Instead, pointers to the LOB data are displayed.

This option can potentially consume a large amount of resources on the local computer.

4. Use the **LOB Saving Option** radio buttons to specify whether you can save large object data (LOB). There are two choices:

• Select **Disable LOB data** to disable saving LOB data on a database data source.

• Select **Enable LOB data** to allow saving LOB data on a database data source.

5. Specify the maximum size of a LOB column in the **Maximum LOB column size** field. You specify the size in kilobytes, up to 2G (the maximum LOB size). The default is 0, which specifies there is no maximum size. If you query a table with LOB data that is larger than the maximum, the LOB data will not be returned for display.

6. Click **Apply** to save your preferences choices.

7. Click **Restore Defaults** to restore the default preference values.

8. Click **OK** to close the Preferences window.

### Setting log preferences

You use the Log page of the Preferences window to activate a log file that will track DataQuant processing information.

### About this task

The log file information can be useful in debugging. It can be delivered as a report or to your workstation in the Output view.

To specify the log file preferences for the DataQuant session:

**Procedure**

1. Select **View > Preferences**. The Preferences window opens. Select **Log** from the tree.

2. Check **Enable log** to activate the log file for debugging purposes and deliver the information in the Output view of your workstation. If this check box is selected, the log file option fields are enabled. Check one or more of the log file options to indicate what information should be gathered. It is recommended that all options remain checked. The available log file options are as follows:

   • Print stack trace for log exceptions
• Print report to log file
• Print log class and method

3. To specify that additional information be gathered, click **Edit Log Categories**. The **Edit Logger Categories** window opens. Select the additional information that should be collected. Click **OK**.

4. Check **Enable report** to deliver the processing information in report format. If this check box is selected, the report file option fields are enabled. Check one or more of the report file options to indicate what information should be gathered. The following check boxes are the report file options:
   • Print stack trace for report exceptions
   • Print report time
   • Print report class and method
   • Automatically activate Output view

5. To specify that additional information be gathered, click **Edit Report Categories** button. The Edit Report Categories window opens. Select the additional information that should be collected. Click **OK**.

6. Click **Apply** to save your preferences choices.
7. Click **Restore Defaults** to restore the default preference values.
8. Click **OK** to close the Preferences window.

### Setting SMTP server preferences

You use the SMTP area of the Preferences window to specify the list of available SMTP servers.

#### About this task

Defining parameters for commonly used SMTP servers in the SMTP area saves users from having to specify these parameters every time they need to send an email message by using these servers. For example, if the SMTP server password is expired, you just need to change it once in the SMTP area instead of updating authorisation information for every object that uses this SMTP server.

The servers listed in the SMTP area can be used in the following procedures:

• Adding a Send Mail action to a Visual Project
• Sending email messages by using the File > Send to command
• Sending email messages by using the MAIL TO procedure command.

To specify SMTP server preferences:

#### Procedure

1. Select **View > Preferences** to open the Preferences window. Select **Email** from the tree and navigate to the **SMTP** area.
2. Add a new server to the list by clicking **Add Server**.
3. In the SMTP window specify SMTP server parameters including its name, address and port number.
4. Optional: If the SMTP server requires authentication, select **Use SMTP authentication**, and then specify user name and password in the **User** and **Password** fields.
5. Optional: If it is necessary to specify additional access preferences, click **Advanced** to open the Advanced SMTP Settings window where you can add required keywords supported by javamail. Click **OK**.
6. Click OK to close the SMTP window and then click Apply to save your preferences choices.

   **Tip:** Select or clear the Enabled check box to control the SMTP settings globally.

7. Optional: If you need to restore default preferences, click Restore Defaults.

8. Click OK to close the Preferences window.

### Setting the directory location preference for objects generated by Export actions

You can set preferences in DataQuant so that objects generated from various DataQuant actions are exported to a specific location on either the local file system or the server file system.

**Before you begin**

If you are using DataQuant for Workstation, create a folder on the local file system.

If you are using DataQuant for WebSphere, create a folder on application server file system.

When you set the **Server-Side File System** preference, you will point to one of these folders.

**About this task**

This task describes how to specify (through preference settings) the location to which generated objects will be exported.

To set the export directory preference:

**Procedure**

1. Select View > Preferences and select Server-side File System from the tree.
2. Select the Enable access to the server file system check box.
3. Type the path in the Root output directory field to indicate where generated objects will be exported.
   When data and objects are generated by the EXPORT procedure, scheduled tasks (Copy or Execute object) or by event actions such as Export to PDF and Export to Excel, they will be exported to path you specify.
   The directory you specify must exist on either the local file system or on the server file system.
4. Type the path in the Root input directory field to indicate the path of the root directory for dynamart source files.
   **Note:** You need to specify a Root input directory path, otherwise the settings are incorrect, and the EXPORT using server-side file system will yield an error.
5. Click Apply to save your preferences.
6. To restore preferences to the default values, click Restore Defaults.
7. Click OK to close the Preferences window.
Setting root directory location for dynamart source files

You can specify a root directory location where dynamart source data files are stored.

About this task

When you save a data file as a dynamart, the link to this file is saved within the dynamart. When you open this dynamart, DataQuant reloads the data from the linked source file and updates the result set if the source file was modified or replaced.

You can specify the Server-Side File System preferences so that the data files that are linked to dynamarts are stored in a specific location on either the local file system or server file system.

Note: You can change the source data file or disable updating of data in the Dynamart Setting window.

Procedure

1. Select View > Preferences and select Server-side File System from the tree of preferences.
2. Select the Enable access to the server file system check box.
3. Type the path of the root directory for dynamart source files in the Root input directory field.

   Note: The directory you specify must exist on either the local file system or on the server file system.

4. Click Apply to save your preferences.
5. To restore preferences to the default values, click Restore Defaults.
6. Click OK to close the Preferences window.

Setting third party library preferences

You use the Third-Party Libraries page of the Preferences window to specify the location of any third-party software that is required by DataQuant.

About this task

To specify the third party library preferences:

Procedure

2. The third-party software whose location has been defined to DataQuant is listed by name, package name and location. In most instances, the location information for third party libraries is specified when DataQuant is installed.
3. To add additional location information for third party software, click Add. The New Third-Party Library window opens. Specify the path to the software in the Path field. If applicable, specify the packages name in the Packages (optional) field. Click OK. The path to the software is added to the Third party library list. Control returns to the Preferences window.
4. To edit the location information for defined third party software, select Edit. The Edit Library dialog opens. You can change the path to the selected software
in the **Path** field. You can change the packages name in the **Packages (optional)** field. Click OK. The changes that you have made are applied. Control returns to the Preferences window.

5. To remove third party library information, click the third party library in the **Third party libraries** list and click **Remove**. The third party library information is removed from the list.

6. Click **Apply** to save your preferences choices.

7. Click **Restore Defaults** to restore the default preference values.

8. Click **OK** to close the Preferences window.

**Specifying Visual Designer preferences**

If you are using DataQuant for Workstation, you use the Visual Designer page of the Preferences window to specify preferences that will effect the creation of visual projects.

**About this task**

To specify the preferences that will effect the creation of visual projects:

**Procedure**

1. Select **View > Preferences**. The Preferences window opens. Select **Visual Designer** from the tree.

2. Click one of the **Visual Designer Ruler Unit** radio buttons to specify the default units that will be used for the ruler that spans the editor window in the **Visual Designer** perspective. Your choices are **pixels**, **inches**, or **centimeters**.

3. Select a formatting schema for date values of a classic report in the **Formatting for &DATE** values drop-down list.

4. Select a formatting schema for time values of a classic report in the **Formatting for &TIME** values drop-down list.

5. Select the **Reset global variables when entering runtime mode** check box to reset the global variable values back to default values each time a visual project is run.

6. Expand **Visual Designer** in the tree and select **Fonts**.

7. To make a different set of fonts available for visual dashboards, specify the path to a custom set of fonts in the **Search directory** field and click **Refresh Associations**.

8. Select **Project Explorer** from the tree.

9. Specify a specific number of times that you can undo an action in the design editor in the **Undo limit** field. A value of '0' indicates that there is no limit to the number of times that you can undo an action.

10. In the **Rename preferences** radio group, specify whether you will be prompted to confirm when you attempt to rename an object in the design editor. The available options include:
   - **Always prompt** - You will always be prompted when you attempt to rename an object.
   - **Prompt when referenced** - You will only be prompted when the object has references to other objects.
   - **Never prompt** - You will never be prompted when you attempt to rename an object.
11. In the **Delete preferences** radio group, specify whether you will be prompted to confirm when you attempt to delete an object in the design editor. The available options include:

- **Always prompt** - You will always be prompted when you attempt to delete an object.
- **Prompt when referenced** - You will only be prompted when the object has references to other objects.
- **Never prompt** - You will never be prompted when you attempt to delete an object.

12. Click **Apply** to save your preferences choices.

13. Click **Restore Defaults** to restore the default preference values.

14. Click **OK** to close the Preferences window.

---

**Exploring the Workspaces view**

All of the data sources and objects that you can access are contained in one or more workspaces that have been pre-populated for you by the DataQuant administrator.

**About this task**

Each workspace to which you have access is listed in the **Workspaces** view. From the **Workspaces** view you can perform the majority of query and reporting functions.

**Discovering the contents of a workspace**

The **Workspaces** view displays the contents of a workspace.

**About this task**

To discover the contents of a workspace:

**Procedure**

1. To open a workspace, click the workspace name in the **Workspaces** view. The content of the workspace is displayed in a tree structure.

2. You can expand any node of the tree by clicking the plus sign (+) to the left of the node name. You continue to expand folders and directories to uncover the contents of the workspace. Many object types are listed in your workspace. The type of object is identified with a unique icon.

3. If you right-click anywhere in the **Workspaces** view, or on an object that is listed in the node, a pop-up menu opens where you can select from one of the available functions.

4. Each data source that has been configured in your workspace is listed and identified with the data source icon. To expand a data source, click the plus sign (+) to the left of the data source name. The Login window opens where you must specify the user ID and password that will be used to login to the data source.

5. Each data source that is listed has one or more of the following folders depending on how the data source was configured:
   - The **Database** folder which includes the native objects (procedures, schemas and tables) that are on the data source.
The **QMF Catalog** folder which lists the objects (forms, queries, procedures, and schemas) that are included in the QMF catalog. This folder is only included if the data source has been configured to access a QMF catalog.

- The **OLAP** folder lists the available OLAP cube models. This folder is only included if the data source has been configured to enable OLAP querying.

6. To open an object that is listed in the **Workspaces** view, double-click that object. Depending on what you have set as your default action, double-clicking on any object listed in your workspace will either run or open the object. The default action for objects is set using **Default Action for Objects** found on the General page of the Preferences window. **Default Action for Objects** control the action for all objects other than visual reports and dashboards.

7. If your user ID has permission, you can add folders to your workspace and move objects into the folders, organizing your information according to your requirements. The DataQuant administrator specifies which users have permission to add folders, move objects and otherwise edit the contents of the workspace.

**Workspace menu commands**

From the **Workspaces** view, you can right-click a node, folder or object and open a pop-up menu. The menu choices depend on what you have selected.

- **New**
  Opens a pop-up menu where you can launch one of the many wizards that are available to help you create objects and other repository items.

- **Open**
  Opens the object that you have selected in the **Workspaces** view. The table is run and the results are returned in the editor window.

- **Open From**
  Opens a menu where you can choose to open an object that has been saved in a:
  - Repository
  - File

- **Open With**
  Opens any database table that is accessible to you in your workspace using a default editor, the Table Editor, the Table Viewer or the Query Editor.

- **Run**
  Runs the object that you have selected in the **Workspaces** view. The results of running the object are returned in the editor window.

- **Run As**
  Runs the visual project that you have selected in the **Workspaces** or **Repository Explorer** views in a specified rendering mode.

- **Set Name Filter**
  Opens the Filter by Name window where you can specify the names of the objects that you want to list for the selected data source in the **Workspaces** view. Using a percent sign (%) will include all objects.

- **Set User Information**
  Opens the Filter by Name window where you can specify the names of the objects that you want to list for the selected data source in the **Workspaces** view. Using a percent sign (%) will include all objects.

- **Rename**
  Renames the object or folder that you have selected in the **Workspaces** view.

- **Delete**
  Deletes the object or folder that you have selected in the **Workspaces** view.
Copy  Copies the object or folder that you have selected in the Workspaces view and writes it to the clipboard.

Paste  Pastes an object or folder from the clipboard to the parent folder or node that you have selected in the Workspaces view.

Paste Link  Pastes the path to the object or folder that is currently on the clipboard into the parent folder or node that you have selected in the Workspaces view.

Add to Favorites  Adds the object that you have selected to the Favorites node in the Personal view.

Add to Startup  Adds the object that you have selected to the Startup node in the Personal view.

Explore  Opens a window in the Editor view that is formatted in an explorer format. Objects, folders, and nodes can be filtered in this window.

Refresh  Refreshes the contents of the Workspaces view that is currently displayed. The Workspaces view is completely refreshed regardless of what node is selected.

Properties  Opens the Properties window for the currently selected object, folder, data source, or node. The Properties window provides specific information about your selection.

Show in Repository  Displays a dialog that shows all objects that reference the selected object in the Repository Explorer view.

Managing workspace folders

If your user ID has permission, you can add folders to your workspace and move objects into the folders, organizing your information according to your requirements.

About this task

The DataQuant administrator specifies which users have permission to add folders, move objects and otherwise edit the contents of the workspace.

To manage repository folders:

Procedure

1. From the Workspaces view, right-click the node or existing folder that you want to be the parent of the new folder. From the pop-up menu, select New > Folder. The Create New Folder window opens.
2. Type the name of the new folder in the Name field.
3. Specify the parent folder of the new folder in the Parent folder field.
4. Optionally, you can type a comment that will describe the folder in the Comment field. You can view the added comment when you view the Properties of the folder.
5. Click Finish to add the folder.

6. To delete a folder, right-click the folder in the Workspaces view. From the pop-up menu, select Delete. The folder and all of its contents are deleted.

7. To rename a folder, right-click the folder in the Workspaces view. From the pop-up menu, select Rename. The folder is renamed.

Filtering data source objects

If you are using DataQuant for Workstation, you can filter by name the objects that will be displayed for a data source in the Workspaces view.

About this task

To filter data source objects:

Procedure

1. From the Workspaces view, right-click a data source that you want to filter. From the pop-up menu, select Set name filter. The Set Name Filter window opens.

2. In the Object owner field, specify the identifier that owns the objects that you want to include in the Workspaces view when the data source is opened. You must specify the owner in the correct format for identifiers. You can use a pattern with wild cards in this field to match multiple owners.

3. In the Object name field, specify the identifier that names the objects that you want to include in the Workspaces view when the data source is opened. You must specify the name in the correct format for identifiers. You can specify a matching pattern to match multiple names.

4. Click OK to close the Set Name Filter window. Control returns to the Workspaces view. Only the objects that meet the criteria that you specified in the Object owner and Object name fields are included.

Creating links to workspace objects

You can set up a link to any object that is listed in the Workspaces view.

About this task

Links are useful if you want to move several objects into a folder without actually copying the object.

To create a link to a workspace object:

Procedure

1. Right-click the object in the Workspaces view. Select Copy from the pop-up menu. The object and the path to the selected object are saved on the clipboard.

2. Right-click the folder that you want to be the parent folder for the link information to the object. Select Paste link from the pop-up menu. The link to the object is listed in the Workspaces view.

3. To delete an object link, right-click the link in the Workspaces view. From the pop-up menu, select Delete. The link is deleted.

4. To rename a link, right-click the link in the Workspaces view. From the pop-up menu, select Rename. The object link is renamed.
Creating links to repository objects

You can set up a link to any object that is in a repository to which you have access.

About this task

Links are useful if you want to access an object without copying the object to your workspace.

To create a link to a repository object:

Procedure

1. Right-click anywhere in the Workspaces view to open the pop-up menu. Select New > Other > Repository > Link. The Create New Link to Repository Object wizard opens.
2. Specify the name for the link information in the Name field. This is the name that will be listed in the Workspaces view.
3. Specify the path including folder name to the parent folder in the Parent folder field. You can use the Browse button to search for the folder.
4. Specify the object for which you want to create a link in the Linked object field. You can use the Browse button to search for the object.
5. Type any text that will be used as comment information in the Comments field. Comment information is displayed in the Properties view for the link.
6. Click Finish. The Create New Link to Repository Object wizard completes its processing.

Viewing properties for workspace objects

The Properties dialog displays the property values for a selected object from the Workspaces view.

About this task

The information provided varies based on the object that is selected. Objects, folders, data sources, and workspaces all have property values that can be displayed. From the Properties dialog you can also modify some property values.

To view or change property values:

Procedure

1. Right-click an item in the repository, select Properties from the pop-up menu. The Properties window opens.
2. The content of the Properties window varies based on the type of item that you select. For all items in the workspace an information page is available. This page gives detailed information on the workspace object. You can add or edit comment information on this page.
3. Additional property pages are available for data sources that are included in the workspace. Select one of the following from the tree:
   • Connection Parameters: To view or edit the connection information that is used to connect to the data source.
   • Plug-ins: To view or change the plug-ins that have been activated for the data source.
   • Resource Limits: To view, define, or edit the resource limits that have been defined for the data source.
- **Login mapping**: To view, define, or edit the login mapping IDs that have been defined for the data source. (DataQuant for Workstation only)
- **Security**: To view, define, or change the users and groups that can access the data source, if it is under the control of the Internal or LDAP security option. You can also view or change the permission levels that have been granted to each user or group. (DataQuant for Workstation only)

4. To restore all the default values for the property values, click **Restore Defaults**.
5. To implement the changes that you have made to any of the property values, click **Apply**.
6. Click **OK** to close the **Properties** window.

### Refreshing workspace contents

You can refresh the contents of the workspace. Refreshing updates any repository objects with the latest versions.

**About this task**

Every time you open an object, the workspace automatically refreshes. You must refresh the workspace if another user saved an object to the same workspace from another session of the application. You can refresh the contents of the **Workspaces** view in one of the following ways:

- Right-click anywhere in the **Workspaces** view. Select **Refresh** from the menu.
- From the **Workspaces** view toolbar, select the **Refresh Workspaces** button.

### Moving workspace objects

If you are using DataQuant for Workstation, using your mouse, you can drag and drop objects and folders that are listed in the **Workspaces** view.

**About this task**

To move workspace objects:

**Procedure**

1. Click an object from the **Workspaces** view.
2. With the mouse button pressed, drag the item to its target location. Release the mouse button. A pop-up menu opens.
3. Select one of the following:
   - **Copy Here** to copy the object to the target location. A copy is made. The original remains.
   - **Move Here** to move the object to the target location. The original copy is moved to the new location.
   - **Create Link Here** to create a link to the object.

### Adding content to a workspace

After you create a workspace, you then add content to the workspace.

**About this task**

If you want to access a data source from the workspace you must add the information necessary to access the data source. If you want objects to be available
from the workspace you must add the information necessary to access the object. For both data sources and objects you can create this information as links or offline entries.

To add content to a workspace:

**Procedure**

1. You can discover data sources and objects that can be added to the workspace in one of the following ways:
   - Open the **Repository Explorer** view. The **Repository Explorer** will display the complete contents of a repository.
   - Open any other workspace to which you have access based on your user ID.

2. Select the data source or object that you want to add to the new workspace. Your user ID must have permission to access the data source or object in order to add it to your new workspace.

3. You can add the data source or object to the workspace in one of the following ways:
   - With the mouse button pressed, drag the data source or object to the new workspace. When you release the mouse button, a context menu opens. Select **Create Link Here** to create a link to the data source or object. Select **Create Offline Entry** to actually copy the data source or object to the workspace.
   - Right-click the data source or the object. Select **Copy** from the pop-up menu. The location of the data source or object is copied to the clipboard. Right-click the workspace to which you want to add the data source or object. From the pop-up menu select **Paste Link** to create a link to the data source or object. Select **Paste as Offline** to actually copy the data source or object to the workspace.

4. Depending on your selection the information required to access a data source or object from the workspace is added as follows:
   - If you selected **Create Link Here** or **Paste Link**, a link to the data source or object is created in the workspace.
   - If you selected **Create Offline Entry** or **Paste as Offline** the data source or object is copied in the workspace. Copying data sources or objects can be time-consuming. However, offline entries allow you to work with an object from a data source without actually accessing the data source. This can save time and cut resource usage.

**Creating additional workspaces**

You can create additional workspaces in the **Workspaces** view if your user ID has been granted full security permission for the repository in which your workspace resides.

The DataQuant administrator assigns a user permission to add workspaces when the repository is created. After creating a new workspace you can add data source information and objects to the workspace.

**Adding a workspace as a user**

If your user ID has full security permission you can add additional workspaces to a repository.
About this task

To add a workspace as a user, perform the following procedure:

Procedure

2. Specify the name for the new workspace in the Name field.
3. Specify a comment that describes the workspace in the Comment field.
4. Click Finish. After you have created a workspace, you specify what data source information and objects will be available in the workspace.

Working in the Personal view

The Personal view provides you with quick access to your most frequently or recently used objects. In addition, it supplies a list of objects that will automatically open when the application starts.

About this task

The Personal view includes the following folders:

- **Recently Used**: This folder contains a list of all the objects recently opened by the user. This list is populated by the application.
- **Favorites**: This folder contains a list of the user's favorite objects. To add an object to the Favorites folder, right-click the object in the Workspaces or Repository Explorer view and select Add to Favorites.
- **Startup**: This folder contains a list of objects that will automatically open upon startup of the User perspective. To add an object to the Startup folder, right-click the object in the Workspaces or Repository Explorer view and select Add to Startup.

Personal view menu commands

From the Personal view, you can right-click a node, folder, or object and open a pop-up menu. The menu choices depend on what you have selected.

- **Remove from Favorites**
  Removes the selected object or folder from the Favorites node.

- **Change favorite name**
  Changes the name of the selected object or folder in the Favorites node. This does not change the original name of the object or folder, only its representation in the Personal view.

- **Remove from Startup**
  Removes the selected object, folder, or node from the Startup node.

- **Change Startup Name**
  Changes the name of the selected object, folder, or node in the Startup node. This does not change the original name of the object, folder, or node; only its representation in the Personal view.
Chapter 3. Creating queries

A query is a request for information from a data source. To request information from a relational data source you can construct a query in a standard fashion by using Structured Query Language (SQL) statements. Or, if you are not familiar with SQL, you can construct a query graphically by using the Query Builder editor.

Query types

DataQuant provides multiple ways to query data. What you can do with queries and the query results can vary depending on the type of query you run.

The following table lists the different query types supported in DataQuant and provides usage information about each type.

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
<th>Characteristics of this type of query</th>
</tr>
</thead>
<tbody>
<tr>
<td>Query</td>
<td>Traditional Query written in SQL or created in the SQL Editor of the Query Builder editor.</td>
<td>• Queries in the QMF Catalog are shared with QMF for TSO and CICS.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Contains SQL and a set of formatting commands for DataQuant for Workstation and DataQuant for WebSphere to execute on the result set.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>These commands are not implemented in QMF for TSO and CICS:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>– grouping and aggregation of column data</td>
</tr>
<tr>
<td></td>
<td></td>
<td>– formatting of the data cells and text</td>
</tr>
<tr>
<td></td>
<td></td>
<td>– creation of Calculated Columns using a legacy syntax from QMF Enterprise Edition V8.x</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Results can be Exported or used with Forms, Visual Reports, and Visual Dashboards.</td>
</tr>
<tr>
<td>Visual Query</td>
<td>A type of Query written in SQL or created with the Query Builder editor.</td>
<td>Saved in the Repository or as a file, NOT saved to the QMF Catalog.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Visual Queries are not shared with QMF for TSO/CICS.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>The Visual Query contains SQL and a set of formatting commands for DataQuant for Workstation and DataQuant for WebSphere family to execute on the result set:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• grouping and aggregation of columns</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• formatting of the data cells and text</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• creation of Calculated Columns using a new more powerful syntax that includes user defined functions in JavaScript</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• instructions to build Charts that can be toggled through by the user</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• filter criteria that act on the result set</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• multilevel prompts for substitution variables</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Results can be Exported or used with Visual Reports, and Visual Dashboards.</td>
</tr>
</tbody>
</table>
Analytical Query

Manipulates results sets of other queries to perform client side joins, appends, crosstabs and other types of analytical transformations.

Analytical queries can use queries and Visual Queries.

Saved in the Repository or as a file, NOT saved to the QMF Catalog.

Not shared with QMF for TSO and CICS.

Contains the SQL from all other queries referenced by it, and a set of formatting commands, identical to those available in the Visual Query, DataQuant for Workstation and DataQuant for WebSphere to execute on the result set.

Results can be Exported or used with Forms, Visual Reports, and Visual Dashboards.

A Query can be converted to a Visual Query with the menu Query>Convert to Visual Query. Query Aggregations of:

Percentage of Group, Percentage of Total, Cumulative Percentage of Group, and Cumulative Percentage of Total are not currently supported by the Visual Query and are omitted upon conversion.

Cloudant Query

An HTTP request that extracts data from a Cloudant database.

Cloudant queries create an index based on the content of a database and retrieve data from the index.

Cloudant queries can be saved to a repository or file and cannot be saved to the QMF catalog.

Cloudant queries are not shared with QMF for TSO/CICS.

Cloudant queries can be used in visual projects.

To work with Cloudant query results (for example, to apply filtering, add a calculated column, create a chart, and so on) a Cloudant query must be added to an analytical query. The results of an analytical query can be managed.

Query interfaces

DataQuant provides robust querying capabilities. You can create new queries using the query building editors or import existing queries. Imported queries can be edited as needed.

In the Query Editor, you can create relational, analytical, and OLAP queries using different query interfaces that are tailored to different skill and knowledge levels. A technical user might select to use a familiar SQL interface, while a business user might opt for a graphical user interface.

In the Cloudant Query Editor, you can create Cloudant queries.

The following table lists the different query editors supported in DataQuant and provides usage information about each editor.

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
<th>Characteristics of this type of editor</th>
</tr>
</thead>
<tbody>
<tr>
<td>SQL Editor</td>
<td>Query is written in SQL.</td>
<td>• Use to create standard and visual queries.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• SQL statements entered directly into editor.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• User should be familiar with SQL.</td>
</tr>
</tbody>
</table>
In the Query Editor, you can switch between the editors by selecting the SQL, Prompted, or Build tabs in the query interface.

**Tip:** Actions taken in one tab automatically appear in the query views shown in the other tabs, allowing users to develop the query using a blend of the three query editors if necessary. However, certain changes made to the query text in the SQL editor (such as unions and subselects) might render the query unsuitable for the Prompted Query or Query Builder editors. In these cases, the query will remain editable in the SQL view only.

You can use the Draw Query wizard to create a query object automatically based on the query type and tables that you specify. This menu option is the same as the DRAW command in QMF for CICS/TSO.

Experienced DataQuant users can use the command bar to access a query or database table. The command bar includes an area for the command itself, along with a means of selecting the server against which the command will run.

### Creating queries using the SQL editor

You can create queries by typing SQL statements in the **SQL Query** editor. The **SQL Query** editor allows you to build standard queries or visual queries.

### Creating queries using SQL

You can create queries by typing SQL statements in the **SQL Query** editor.

**About this task**

This task describes how to create queries by using **SQL Query**. It assumes that you are familiar with SQL and SQL syntax.

You can write a single SQL statement that will return a single result set or multiple SQL statements that will return multiple result sets. The **SQL Query** editor provides coloring support for your SQL statements.

To create a query using the **SQL Query** editor:
Procedure

1. Select File > New > Query. The Create New Query wizard opens. Specify the name of the query in the Name field and where the query will be saved in the Data Source field. Click Finish. Click the SQL tab.

2. Type the text of the SQL statement in the input text area. You are responsible for ensuring that the SQL statements that you type are syntactically correct.

3. You can type multiple SQL statements in a single query window. Each statement will run and multiple result sets (one for each statement) will be returned. You must include a semicolon (;) between each SQL statement.

Note: If you are using MySQL V5.1 you can have a valid SELECT statement without tables being specified or with a pseudo table DUAL and the Add New button for Columns is always enabled in the Prompted tab.

DataQuant Version 1.2 Fix Pack 16 supports SQL syntax for the MySQL V5.1 database.

The following MySQL attributes (keywords) in the SELECT statement are supported:

Note: For the MySQL V5.1 database, these keywords are supported by but cannot be viewed or edited by using the Prompted or Diagram query editors. These keywords can be viewed and edited in SQL tab only:

- HIGH_PRIORITY
- STRAIGHT_JOIN
- SQL_SMALL_RESULT / SQL_BIG_RESULT / SQL_BUFFER_RESULT
- SQL_CACHE / SQL_NO_CACHE
- SQL_CALC_FOUND_ROWS
- LOCK IN SHARE MODE
- LIMIT rowcount

The general syntax of the SELECT statement is:

```
SELECT [HIGH_PRIORITY] [STRAIGHT_JOIN] [SQL_SMALL_RESULT | SQL_BIG_RESULT | SQL_BUFFER_RESULT] [SQL_CACHE | SQL_NO_CACHE] [SQL_CALC_FOUND_ROWS] select clause [FROM table clause [WHERE clause] [GROUP BY clause] [HAVING clause] [ORDER BY clause] [LIMIT clause] [FOR UPDATE | LOCK IN SHARE MODE]]
```

For example: SELECT 1 + 2 is a valid SELECT statement

Additionally, the following clauses are supported but cannot be viewed or edited in the Prompted or Diagram query editors directly; they can be viewed or edited in SQL tab only:
Creating visual queries using SQL

You can create visual query objects in DataQuant.

About this task

This task describes how to create visual queries using SQL.

To create visual queries using SQL:

Procedure

1. Open the Create New Visual Query wizard by selecting File > New > Other and expanding the DataQuant Objects folder or by clicking the New Visual Query ( ) icon on the toolbar.
2. Select Visual Query from the list of DataQuant Objects.
3. Specify the name of the visual query in the Name field and where the visual query will be saved in the Data Source field.
4. Click Finish.
   The Query editor opens.
5. Click the SQL tab to open the SQL input area in the editor.
6. Type the text of the SQL statement in the input text area.
   You are responsible for ensuring that the SQL statements are syntactically correct.
7. Click the Run Query ( ) icon on the toolbar.
   The visual query is run against the currently connected to data source. Results are returned to the interface. The query results are displayed in the Results tab. You can switch between the multiple views by clicking the corresponding tabs.
8. Once you have obtained query results you have several options.
• Format the columns and rows of the query results
• Aggregate columns or rows
• Add calculated columns to the query results
• Define variables using a substitution window
• Generate charts
• Generate quick reports
• Export the query results to numerous formats.

Note: The SQL visual query object stays open until you close it. Upon closing, if you want to run the visual query object again you must save it to repository.

9. Click the Save icon to open the Save object window. You can save the visual query object to a file or the repository.

Using Content Assist for queries and visual queries
If you are using DataQuant for Workstation, Content Assist helps to complete SQL statements by providing lists of options while the statement is being typed.

About this task
To use Content Assist while typing a SQL statement:

Procedure
1. Depending on whether you are creating a query or a visual query, perform one of the following steps:
   a. For queries, select File > New > Query. The Create New Query wizard opens. Specify the name of the query in the Name field and where the query will be saved in the Data Source field. Click Finish. Click the SQL tab.
   b. For visual queries, select File > New > Other and expand the DataQuant Objects folder. Select Visual Query from the list of DataQuant Objects.
2. Specify the name of the query in the Name field and where the query will be saved in the Data Source field. Click Finish.
3. Click the SQL tab.
4. At any time while typing the SQL statement, press Ctrl+Spacebar. A drop-down list opens underneath the cursor. The contents of the drop-down list depend on the location of the cursor in the SQL statement and the contents of the table being called. For example, if SELECT * FROM Q is displayed when Content Assist is selected, the drop-down list will display all tables in the data source with the owner name Q. If SELECT * FROM Q.STAFF WHERE is displayed, the drop-down list will display all columns of the Q.STAFF table.
5. If you select Content Assist while partially typing a column, function, or table reference, and the partially typed keyword is not ambiguous, Content Assist will automatically complete the keyword. For example, when typing SELECT * FROM Q.ST, the STAFF table is automatically completed since it is the only table under the Q schema that begins with the letters ST.
6. Select an option from the drop-down list. The selected text is automatically inserted into the SQL statement.

Using Parameter Hints for queries and visual queries
If you are using DataQuant for Workstation, Parameter Hints provide a summary of a function’s parameters as it is being typed in SQL statements.
About this task

To use Parameter Hints while typing parameterized functions into SQL text:

Procedure

1. Depending on whether you are creating a query or a visual query, perform one of the following steps:
   a. For queries, select **File > New > Query**. The Create New Query wizard opens. Specify the name of the query in the **Name** field and where the query will be saved in the **Data Source** field. Click **Finish**. Click the **SQL** tab.
   
b. For visual queries, select **File > New > Other** and expand the **DataQuant Objects** folder. Select **Visual Query** from the list of DataQuant Objects.

2. Specify the name of the query in the **Name** field and where the query will be saved in the **Data Source** field. Click **Finish**.

3. Click the **SQL** tab.

4. Type the first part of a parameterized function, which includes the name of the function and the left parenthesis. For example, `SUM(`.

5. Press **Ctrl+Shift+Spacebar**. A ToolTip opens that displays an example of how to complete the function. For example, if `SUM(` is displayed, the ToolTip will display `SUM(<numeric-expression>).`

6. Use the displayed hint as a reference when completing the parameterized function.

---

Creating queries using the Prompted Query editor

You can build simple to complex queries using the **Prompted Query** editor. The **Prompted Query** editor prompts you for the information it needs to build the SQL statement.

About this task

To create a new SQL query using the **Prompted Query** editor:

Procedure

1. Select **File > New > Query** to open the Create Query wizard.
   
a. Specify the name of the query in the **Name** field and specify where the query will be saved in the **Data Source** field.
   
b. Click **Finish**.
   
c. Click the **Prompted** tab.

2. Use each of the following panes to build your query:
   
   - **Tables** where you specify the tables that will be used in the query.

     If you are running a version of DB2 that supports temporal data (for example, DB2 z/OS v10), when a table that has been configured for temporal data is added to the prompted query, the **Time Period** icon is enabled. If there are multiple tables in the query, the **Time Period** icon is enabled only when the table that is configured for temporal support is selected. Clicking the **Time Period** icon launches a window in which you can specify the points in time as either system time or business time (depending on the type of time for which the table has been configured). Only one of the system time or business time pair will be enabled for any given table.


Temporal data support provides a historical perspective to table data. Historical data can often serve a specific business case. For example, a business might require access to information about the level of insurance coverage that was in place at the time of a claim several months in the past. A table with temporal data support stores and records information about the period of time when a row is valid with respect to system time, business time, or both. System time, or transaction time, is when the transaction is recorded. Business time, or valid time, is when the data is valid with respect to information about the current real world business data.

If the DBA specified the tables and columns that require temporal data support when they are created, DB2 automatically maintains the history whenever an update is made to the data.

To specify a time period, click the **Time Period** icon and specify values for **System Time** and **Business Time**.

a. The drop-down list to the right of the **For System Time** and **For Business Time** labels provide SQL reference options.

b. Select **As Of**, **From**, or **Between**. Consult your SQL reference documentation for details on the usage of these options.

c. Set the associated date by clicking the date and time icon.

If you do not need the period you have specified, simply clear the corresponding check box. If you remove a table or edit its owner or name, then all the specified time periods will be lost.

The **As Of** option creates a temporal specification in the SQL of a FOR statement `<SYSTEM_TIME | BUSINESS_TIME> AS OF <selected timestamp>`

The **From** option creates a temporal specification in the SQL of a FOR statement `<SYSTEM_TIME | BUSINESS_TIME> FROM <selected timestamp>`

The **Between** option creates a temporal specification in the SQL of a FOR statement `<SYSTEM_TIME | BUSINESS_TIME> BETWEEN <selected timestamp1> AND <selected timestamp2>`

- **Join conditions** where you specify the join conditions for tables if you are using multiple tables in the query.
- **Columns** where you specify the columns that will be included in the query results.
- **Sort conditions** where you specify how the rows in the query results will be sorted.
- **Row conditions** where you specify conditions that will limit the rows that are returned in the query results.
- **Include duplicate rows** check box where you specify that duplicate rows will be included in the query results.

3. When you are done building the query, click the **Run Query** icon.

The query is run against the currently connected to data source. Results are returned.

The query results are identified with a **Results** tab.

You can switch between the multiple views of a query (SQL, Prompted, Build, Layout, Preview, and Results) by clicking the corresponding tabs.

Once you have obtained query results you have several options. You can:

- Format the query results’ columns and rows.
- Aggregate columns or rows.
- Add calculated columns to the query results.
• Generate reports
• Export the query results to numerous formats.

The query object stays open until you close it. Upon closing, if you want to run the query object again you must save it.

4. Click the Save icon to open the Save object window.

Depending on the permissions granted to your user ID, you can save the query object to a file, a workspace, the QMF catalog, or the repository.

Adding tables

You use the Tables window to add a table to a query when you are building the query using the Prompted Query editor. You can also use this dialog to change the tables that have been specified in the query.

About this task

To add a table to a query when you are building the query using the Prompted Query:

Procedure

1. Open the Tables window from the Prompted Query editor.
2. Identify the table that you want to include in the query. Type the table owner in the Table owner field.
3. Click the Add Table toolbar button in the Tables pane.
4. Type the table name in the Table name field.
5. If you do not know the table owner and name, you can search the data source to see what tables are available. To search for a table:
   a. Specify the owner identifier of the tables that you want to include in the list in the Table Owner field. You must specify the owner in the correct format for identifiers. You can specify a matching pattern to match multiple names. To list all tables use the % sign.
   b. You can further narrow the tables listed by using the name field. Specify an identifier for the table names that you want to include in the list in the Table Name field. You must specify the name in the correct format for identifiers. You can specify a matching pattern to match multiple names. To list all tables for a particular owner use the % sign.
   c. Click the Add From List button to search the data source for tables that match the criteria that you specified. The Table List window opens listing all the tables on the data source that met your search criteria.
   d. Select a table to include in the query from the list of tables.
6. If you want to use the specified owner name for each successive added query, select the Save Owner Filter check box. Each time that you open the Tables dialog, box the saved owner name will automatically be displayed in the Table owner field.
7. Click Add. The table is added to the query. The name of the table appears in the Tables pane of the Prompted Query editor.

Joining tables

You can add more than one table. When you add an additional table to a query you must specify how the new table will be joined to an existing table.
About this task

When you add an additional table to a query using the Tables window, the Join Conditions dialogs open automatically.

There are two Join Conditions dialogs:
- The Join Tables window, where you specify what type of join will link the tables, such as an inner join or left, right or full outer join.
- The Join Columns window, where you specify the columns from each table that will be joined.

To join a table:

Procedure
1. Open the Join Tables window using the Prompted Query editor.
2. Select the table that is to be joined to the original table from the Select a table to join into the query list of tables.
3. Select the type of join that will link the two tables. Select the radio button for one of the following types of joins:
   - **Inner join** to include in the query results only those rows with matching values in both tables. An **Inner join** is implicit if you do not specify any other join operator. An **Inner join** compares every row of the left table with every row of the right table keeping only the rows that match. The resulting table might be missing rows from both of the joined tables.
   - **Left outer join** to include in the query results all the rows in the left table and only the rows from the right table that match with rows from the left table.
   - **Right outer join** to include in the query results all the rows in the right table and only the rows from the left table that match with rows from the right table.
   - **Full outer join** to include in the query results all rows from both tables.
4. Click Continue to complete the joining tables process.

What to do next

The step to joining tables is joining columns.

Joining columns

You must specify the columns that will be used to join the tables.

About this task

The Join Columns window opens automatically as the second phase to specifying join conditions for multiple tables when you are building a query using the Prompted Query editor. Rows in each of the tables that have equal values in these columns will be joined.

To join columns:

Procedure
1. Open the Join Columns window using the Prompted Query editor.
2. Select one column from the Available columns to join list.
The columns from all the tables that have been included in the query are listed. You must select a column with the same data type (NUMERIC, DATE, TIME) as the column selected in the Columns of (table name) list.

3. Select one column from the Columns of (table name) list.
   Only the columns from the right table (the table that you are joining) are listed. You must select a column with the same data type (NUMERIC, DATE, TIME, CHARACTER) as the column selected in the Available columns to join list.

4. Click Add.
   Rows that have equal values in the columns that are listed in the Available columns to join list and the columns that are listed in the Columns of (table name) list will be joined.
   The Join Columns window closes.

What to do next

The next step is to specify what columns will be in the query results.

Specifying columns for query results

You can specify the columns that are included in the query results.

About this task

You use the Columns window to specify what columns are included in the query results in the Prompted Query or Query Builder editor. By default, all the columns from a table that is included in the query are included in the query results. If you have multiple tables included in the query, all the columns from each table are included.

To specify the columns that are included in the query results:

Procedure

1. Open the Columns window by clicking the Add Column button.
   • Each table that was added to the query is listed in the Table field.
     If there are two or more tables, each table is prefixed by a letter, such as Q.STAFF(A) and Q.INTERVIEW(B).
   • All the table columns are listed in the Column field.
   • The type of data that is contained in the column is listed in the Type field.
   • The label that is associated with the column is listed in the Label field.
     Labels for columns are system column headers or column text. They are used when the query is displayed or when the results are printed.
   • Any comments that are associated with a column are listed in the Comments field.

2. You can include a column in the query results in one of the following ways:
   a. Select a column from the list of available columns by clicking the name of the column as it appears in the Column field.
      You can filter the listed columns by typing a necessary column name or label, which is associated with the column, in the text field in the Value area.
   b. Select all the columns of a table by selecting the name of the table in the Table field.
c. Select all the columns of a table by selecting the value `<All columns>` in the Column field.

d. Enter an expression in the **Or, enter an expression here** field.
   The expression that you enter is used to evaluate a column and determine whether it is included in the query results.

   **Tip:** Click the ellipsis (...) to open an expression builder. The expression builder offers a palette of common elements that are used to create SQL expressions, such as column names, constants, functions, and operators. When you click the expression builder buttons, templates for expression elements are inserted into the expression fields.

3. From the **Summary function** list, select how you want to summarize the column in the query results.
   You can specify the following summary options:
   - **(None)** - No summarization is applied.
   - **Average** - The average of all values in the column is included in summary field.
   - **Count** - A count of all the rows is included in the summary field.
   - **Maximum** - The maximum value that was in the column is included in the summary field.
   - **Minimum** - The minimum value that was in the column is included in the summary field.
   - **Sum** - The sum of all the values in the column is included in the summary field.

4. To specify a new name for the column in the query results, type the new name in the **New column name** field.
   Unless you specify a new name, the name of the column in the column header of the query results is displayed exactly as it appears in the Column field.

5. Click **Add**.
   The column is included in the query results. Repeat these steps for each column that you want to include in the query results.

6. If you work in the **Prompted Query** editor, click the **Change** button if you want to change to a column that was added to the query results.
   The change to the column is made, and the Columns window closes.

7. Click **Close** when you finish adding all the columns that you want to be included in the query results. The Columns window closes.

### Specifying sort conditions for query results

You can specify the sort conditions that determine the order that the rows are returned in the query results.

**About this task**

Use the Sort Conditions dialog to specify the sort conditions that will apply to the query results. The Sort Conditions dialog is used when building a query using the Prompted Query editor. Rows can be sorted in ascending (A-Z) or descending (Z-A) order. If you sort query results rows by more than one column, the first column is ordered first, then the second column is ordered within the sort order defined for the first column.

To specify the sort conditions for the query results:
Procedure

1. Open the Sort Conditions window by clicking the Add Sort Condition button in the Sort conditions pane of the Prompted Query editor.

2. Specify the columns that will be used to sort the query results in one of the following ways:
   - Select one or more columns from the Columns in the result set list. The Columns in the result set lists the columns that are included in the query results. You can select one or more columns if you are adding a sort condition. You can select only one column if you are changing a sort condition.
   - Specify a conditional expression in the Or, enter a condition here field. Click the ellipsis (...) to open an expression builder that will help you build the conditional expression. The expression builder offers a palette of common elements used to create SQL expressions, such as column names, constants, functions, and operators. When you click the expression builder buttons, templates for expression elements are inserted into the expression fields.

3. Specify the sort direction.
   - Click Ascending to specify that the query results will be sorted in ascending (lowest-to-highest) order.
   - Click Descending to specify that the query results will be sorted in descending (highest-to-lowest) order.
   - The first column is used for the primary sort; subsequent columns will sort within the first.

4. Click Add to add the sort condition to the query results.
   - If you are changing a sort condition, click Change.

   Repeat this process for each column that you want included in the sort conditions.

   When you have finished selecting the columns that will be included in the sort conditions, click Close. The Sort Conditions window closes.

Specifying row conditions for query results

You can specify row conditions to limit the rows that are returned in the query results. Without row conditions, all qualifying rows will be returned from the query.

About this task

You use the Row Conditions dialog to specify the row conditions for the query results. The Row Conditions dialog is used when building a query using the Prompted Query editor.

To specify the row conditions for query results:

Procedure

1. Open the Row Conditions window by clicking Add Row Condition in the Row conditions pane of the Prompted Query editor.

2. If you are specifying more than one condition that must be met for a row to be included in the query results, you must indicate how each condition will be connected using the Connector radio buttons.

   • If you select And, then the current condition that you are specifying and the previous condition (listed in the Row conditions pane of the Prompted Query editor) must be met in order for the row to be included.
If you select **Or**, then the current condition or the previous condition can be met in order for the row to be included.

When you connect multiple conditions using both **And** and **Or**, then the rules of the data source determine how the conditions are processed.

3. Select a column from the **Select a column from the list** to examine for inclusion in the query results.

The **Select a column from the list** lists all the columns that will be included in the query results.

You can filter the listed columns by typing a necessary row name in the text field in the **Left Side** area.

4. To specify a conditional expression that will determine the column, enter an expression in the **Or, enter an expression here** field.

5. Click the ellipsis (...) to open an expression builder that will help you build the conditional expression.

The expression builder offers a palette of common elements used to create SQL expressions, such as column names, constants, functions, and operators.

When you click the expression builder buttons, templates for expression elements are inserted into the expression fields.

6. Specify the operator for the row condition expression using the **Operator** field.
   a. To use the operator as is, select **Is**.

   For example, to specify "Is Not Equal To" click **Is Not** and select the **Equal To** operator in the list. You cannot use the **Starting with**, **Ending with**, or **Containing** operators if the left and right sides of the row condition expression have numeric data types.

   **Note:** If IBM DB2 Spatial Extender is installed on the database, you are able to work with spatial data types: points, lines, and polygons. But you cannot use any of the spatial predicates unless the left and right sides have spatial data types.

7. Specify the condition for which to check the rows in the **Right Side** area.
   a. You can use the expression builder button to help you build the conditions.

   b. To add row conditions from a file of any format with the values that are separated by commas, click **Import Conditions From File**.

   c. To specify row conditions from a set of query results, click **Import Conditions via Executing SQL Query**, and enter the SQL text of a query.

   d. You can use the **Remove Condition** or **Remove All Conditions** buttons to remove one or all row conditions.

8. Click **Add**.

   The row condition is added and displayed in the **Row conditions** pane of the **Prompted Query** editor.

   Repeat this process for each condition that you want to add.

9. When you have finished specifying each condition, click **Close**.

   The Row Conditions window closes and control returns to the **Prompted Query** editor.
Creating queries and visual queries in the Query Builder editor

You can use the Query Builder editor to create queries and visual queries. You supply table, join, column, sort, and row information and the editor constructs the SQL statements.

About this task

To create a query or a visual query in the Query Builder editor, you must add tables. Then you can add joins to them, to specify particular table fields, add row and global conditions, or to edit the SQL text of the created query:

Adding and joining tables in the Query Builder editor

To create a query or a visual query in the Query Builder editor, you must add tables first.

About this task

To add tables in the Query Builder editor:

Procedure

1. From the main menu, select File > New > Query. The Create New Query wizard opens. Specify the name of the query in the Name field and the data source that you want to use for the query in the Data Source field. Click Finish.
2. Click the Build tab. The Query Builder editor opens. The Build tab is divided into three sections: Tables, Fields, and SQL Editor.

Tip: Expand and collapse the sections by clicking buttons next to the section names to make all functions of a particular section available.

3. To add a table to the query, click the Plus button. In the Tables window, select one or more tables to include in the query. A visual representation of each table that you add appears in the Tables section. The Suggested Tables list contains the tables from ER diagrams of the specified data source. To add a table from Suggested Tables, double-click it.
4. To remove a table from the query, navigate to the table that you want to remove and click Close.
5. To sort columns in the table, navigate to the table the columns of which you want to sort and click the sorting button. You can also clear the Show All Columns check box to hide the columns that are not included in the query.
6. To specify relationships among several tables and join tables, select a column from one table and drag it to a column of another table. You can set up multiple join conditions.

Note: If the Tables section is expanded, a connecting line is drawn from the column in one table to the column in the second one.

7. Optional: To edit the type of join conditions, in the expanded Tables section, right-click the join line that appears between any two tables. Select Change Join from the menu to change the join conditions. The Join Tables window opens. You can select new join conditions. For detailed information about types of join conditions, see the step 2 in the Editing join conditions for multiple tables topic.

Chapter 3. Creating queries
8. To remove the join condition, right-click the join that you want to delete and select Remove Join.

**Editing join conditions for tables**

You can edit the join conditions to alter how two tables are linked.

**About this task**

To edit the join conditions that have been specified for two tables:

**Procedure**

1. Right-click the line that joins the table diagrams in the **Tables** section of the Query Builder editor window. Click **Change Join**. The Join Tables window opens.
2. Select the type of join that will link the two tables from one of the following choices:
   - **Inner join** to include in the query results only those rows with matching values in both tables.
   - **Left outer join** to include in the query results all the rows in the left table and only the rows from the right table that match with rows from the left table.
   - **Right outer join** to include in the query results all the rows in the right table and only the rows from the left table that match with rows from the right table.
   - **Full outer join** to include in the query results all rows from both tables.
3. Click **Change** to save the new join conditions.

**Specifying fields and conditions for a query or a visual query**

In the **Fields** section of the Query Builder editor, you can select the columns that you want to include in the query result set and specify the sort and row conditions for the query results.

**About this task**

To specify table fields in the Query Builder editor:

**Procedure**

1. Expand the **Fields** section on the **Build** tab.
   * **Note:** All columns of all the tables that are selected for the query are automatically included in the query results column table. The <All columns> entry displays when all table columns are included in the query.
   - To display particular columns in the query results, you can select or clear the corresponding check boxes.
   - To add individual columns to the query results, drag a column from one of the tables in the **Tables** section to a particular row in the query results table in the **Fields** section.
2. To specify particular columns to a query, see the "Specifying columns for query results" on page 39 topic.
3. For each column that you included in the column table, you can apply sort and row conditions in the corresponding table fields. Click one of the cells in the column to specify a value for one or more of the following columns:
<table>
<thead>
<tr>
<th>Column Name</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table</td>
<td>The column specifies the name of the table that includes the current column.</td>
</tr>
<tr>
<td>Column Name</td>
<td>The column specifies the name of the current column. <strong>Tip:</strong> Double-click the Column Name cell, and then click the down arrow. All column names from the current tables are listed. You can select another column name to replace the current column or remove it.</td>
</tr>
<tr>
<td>Column Type</td>
<td>The column displays the type of data that the current column contains.</td>
</tr>
<tr>
<td>Display Name</td>
<td>The column specifies the name that is used as the column header in the query results.</td>
</tr>
<tr>
<td>Aggregation</td>
<td>The column specifies the type of aggregation that is applied to the column. Double-click the Aggregation cell, and click the down arrow.</td>
</tr>
<tr>
<td>Sort Order</td>
<td>The column specifies whether this column is used to sort the rows in the query results. Double-click the Sort Order cell, click the down arrow, and select the sorting.</td>
</tr>
<tr>
<td>Sort Sequence</td>
<td>The column specifies the sort order sequence for the column. The column with the lowest number is sorted first.</td>
</tr>
<tr>
<td>Conditions</td>
<td>The column specifies a condition that must be met for a row of data to be included in the query results. The condition that you specify in this column applies to the column value. For example, if you had a column that is called Number and you entered a row condition of &gt; 10, the only rows of data that are included in the query results are those rows that have a value greater than 10 in the Number column. Type the row conditions that you want to apply to each column.</td>
</tr>
</tbody>
</table>

4. To specify whether to add duplicate rows to the query results, select the **Include duplicate rows** check box.

**Specifying columns for query results**
You can specify the columns that are included in the query results.

**About this task**

You use the Columns window to specify what columns are included in the query results in the **Prompted Query** or **Query Builder** editor. By default, all the columns from a table that is included in the query are included in the query results. If you have multiple tables included in the query, all the columns from each table are included.

To specify the columns that are included in the query results:

**Procedure**
1. Open the Columns window by clicking the **Add Column** button.
   - Each table that was added to the query is listed in the **Table** field.
     - If there are two or more tables, each table is prefixed by a letter, such as **Q.STAFF(A)** and **Q.INTERVIEW(B)**.
   - All the table columns are listed in the **Column** field.
• The type of data that is contained in the column is listed in the **Type** field.
• The label that is associated with the column is listed in the **Label** field. Labels for columns are system column headers or column text. They are used when the query is displayed or when the results are printed.
• Any comments that are associated with a column are listed in the **Comments** field.

2. You can include a column in the query results in one of the following ways:
   a. Select a column from the list of available columns by clicking the name of the column as it appears in the **Column** field. You can filter the listed columns by typing a necessary column name or label, which is associated with the column, in the text field in the **Value** area.
   b. Select all the columns of a table by selecting the name of the table in the **Table** field.
   c. Select all the columns of a table by selecting the value `<All columns>` in the **Column** field.
   d. Enter an expression in the **Or, enter an expression here** field. The expression that you enter is used to evaluate a column and determine whether it is included in the query results.

   **Tip:** Click the ellipsis (...) to open an expression builder. The expression builder offers a palette of common elements that are used to create SQL expressions, such as column names, constants, functions, and operators. When you click the expression builder buttons, templates for expression elements are inserted into the expression fields.

3. From the **Summary function** list, select how you want to summarize the column in the query results.
   You can specify the following summary options:
   • (None) - No summarization is applied.
   • **Average** - The average of all values in the column is included in summary field.
   • **Count** - A count of all the rows is included in the summary field.
   • **Maximum** - The maximum value that was in the column is included in the summary field.
   • **Minimum** - The minimum value that was in the column is included in the summary field.
   • **Sum** - The sum of all the values in the column is included in the summary field.

4. To specify a new name for the column in the query results, type the new name in the **New column name** field. Unless you specify a new name, the name of the column in the column header of the query results is displayed exactly as it appears in the **Column** field.

5. Click **Add**. The column is included in the query results. Repeat these steps for each column that you want to include in the query results.

6. If you work in the **Prompted Query** editor, click the **Change** button if you want to change to a column that was added to the query results. The change to the column is made, and the Columns window closes.

7. Click **Close** when you finish adding all the columns that you want to be included in the query results. The Columns window closes.
Specifying global row conditions

In the Query Builder editor, you can specify global row conditions for a query or a visual query.

About this task

The global row conditions affect more than a specific column in the query results. You can use this pane if you want to set a condition that includes the data column that is available in the file, but is not included in the query results.

To specify global row conditions:

Procedure

1. Click Show Row Conditions and open the Global Conditions pane.
2. Click Insert Condition. The Row Conditions window opens.
3. If you specify more than one condition that must be met for a row to be included in the query results, you must indicate how each condition is connected in the Connector area.
   - And connector means that the current condition and the previous condition must be met in order for the row to be included.
   - Or connector means that either the current condition or the previous condition can be met in order for the row to be included.

Tip: When you connect multiple conditions with both And and Or, the rules of the data source determine how the conditions are processed.

4. In the Left Side area, select a column from the list that you want to use in the conditional expression for adding to the query results.
   This list contains all the columns that are included in the query results. You can filter the listed columns by typing a necessary row name in the text field.
5. If you want to specify a conditional expression that determines the column, enter an expression in the Or, enter an expression here field.
   a. Click the ellipsis (...) to open an expression builder that can help you build the conditional expression. The expression builder offers a palette of common elements that are used to create SQL expressions, such as column names, constants, functions, and operators.
   b. Click the expression builder buttons, templates for expression elements are inserted into the expression fields.
6. Select the operator for the global row condition expression in the Operator area.
   a. To use the operator as is, select Is.
   b. To change the operator to its negative opposite, click Is Not.
      For example, to specify "Is Not Equal To" click Is Not and select the Equal To operator from the list.

Important: You cannot use the Starting with, Ending with, or Containing operators if the left and right sides of the row condition expression have numeric data types.

Important: If IBM DB2 Spatial Extender is installed on the database, you are able to work with spatial data types: points, lines, and polygons. But you cannot use any of the spatial predicates unless the left and right sides have spatial data types.
7. Specify the condition for which to check the rows in the Right Side area.
   a. You can use the expression builder button to help you build the conditions.
   b. To add row conditions from a file of any format with the values that are separated by commas, click Import Conditions From File.
   c. To specify row conditions from a set of query results, click Import Conditions via Executing SQL Query, and enter the SQL text of a query.
   d. You can use the Remove Condition or Remove All Conditions buttons to remove one or all row conditions.

8. Click Add. The row condition is added and displayed in the Global Conditions pane of the Query Builder editor. Repeat this process for each condition that you want to add.

9. When you finish specifying each condition, click Close.

10. In the Global Conditions pane you can do the following actions:
    a. To change the order of the entered conditions, select a condition from the list and click Move Condition Up or Move Condition Down.
    b. To insert a condition before a particular condition, navigate to the required condition, click Insert Condition, and enter the text of the condition in a blank line. The condition is added to the list before the selected one.
    c. To edit a command, select it from the list, click Edit Condition, and change the text of the condition.
    d. To remove a condition, select it from the list, click Remove Condition or delete the text of the condition from the line.

**Editing SQL text**

The changes that you make in the Tables and Fields sections are automatically applied in the SQL Editor section to the query or a visual query.

**About this task**

To edit the SQL text of a query in the Query Builder editor:

**Procedure**

1. Expand the SQL Editor section on the Build tab.
2. Edit the SQL text of a query or a visual query.
3. Click the Run Query toolbar button. The query is run against the currently connected data source. The query results are displayed on the Results tab.

**Creating queries using the Draw Query wizard**

Use the Draw Query wizard to create a query object automatically based on the query type and tables that you specify. This menu option is the same as the DRAW command in QMF for CICS/TSO.

**About this task**

The Draw Query wizard builds the SQL statements for the query.

To create a query using the Draw Query wizard:
Procedure

1. Open the Create New Query Using the Draw Query wizard by clicking the Draw Query toolbar button.
   You can also open the Create New Query Using the Draw Query wizard by selecting File > New > Other > Query using Draw Query wizard.
2. From the Data Source list select the data source where the tables that you want to use in the query are stored.
3. Click one of the Query Type radio buttons to select the type of query that you want to create. You can select one of the following:
   - Select to create a SQL SELECT statement that will retrieve rows of data from one or more tables.
   - Update to create a SQL UPDATE statement that will allow you to change data that is contained in a table.
   - Insert to create a SQL INSERT statement that will add new rows to a table. You can only update one table at a time using this statement.
4. In the Query Tables list specify the table or tables that will be included in the query.
   a. To specify a table, type the owner of a table in the Owner field, the name of the table in the Name field, and the correlation ID of the table in the Identifier field.
   b. Click Add.
   The table is listed in the Tables list.
5. If you do not know the table name, you can search a list of tables that are available on the data source, by clicking Add from list.
   The Object List window opens. Using the Object List window you can select one or more tables to include in the query if you have chosen the Select option as the query type.
6. To remove a table from the Tables list, select the table name and click the Remove Table icon. Use the Move Table Up in List or the Move table Down in List icons to move a table up and down in the Tables list.
7. Click Finish to create the query.
   The Create New Query Using Draw Query Wizard closes. The SQL for the query is displayed in the query workstation window. You can edit the SQL in the available window.

Listing data source tables

Use the Objects List dialog box to list the tables that are available on the data source.

About this task

To list the tables on a data source:

Procedure

1. To open the Objects List dialog, box click the Add from list button that appears in the Draw Query wizard.
2. Enter the owner of the table that you want to list in the Owner field and enter the name of the table that you want to list in the Name field.
   You must specify the name in the correct format for identifiers. You can specify a matching pattern to match multiple names. Use the percent character (%) to match a string of any length containing any characters. For example, to list all
tables with a name beginning with the letter A, enter A%. Use the underscore character (_) to match a single character. For example, to list all tables with an owner that has the letter A in the second position, enter _A%.

3. Click the Refresh List button to initiate a search for the tables on the database that match the search criteria that you specified in the Owner and Name fields. Only those tables that you are authorized to use are included in the list. The results are returned in the Object List list box. From this list select the table or tables that you want to use in your query.

4. Click OK to close the Object List dialog box.
Control returns to the Draw Query wizard. The table or tables that you selected are listed in the Tables list.

---

### Working with prompts in queries

You can filter your queries or dynamarts by adding prompts that direct the query or dynamart to focus on specific values or ranges. This limits the amount of data to be queried and provides more focused query results. The filtering works on the database level.

**About this task**

Use the following types of prompts depending on the object type:

<table>
<thead>
<tr>
<th>Table 6. Prompt types</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Object type</strong></td>
</tr>
<tr>
<td>Query</td>
</tr>
<tr>
<td>Visual query, dynamart</td>
</tr>
<tr>
<td>Analytical query, compound dynamart</td>
</tr>
</tbody>
</table>

The following topics contain the detailed information about adding prompts:

**Working with SQL prompts**

You use SQL prompts to limit the data retrieved from a query, visual query, or dynamart.

**About this task**

SQL prompts are part of SQL statements that you add when you write a condition to select only the data that is filtered by a specified value. When you run a query, you must specify the value for this prompt before you get the result set.

To add an SQL prompt to a query or a visual query:

**Procedure**

1. Open the SQL tab.

   **Note:** You can specify a prompt in the Conditions column on the Build tab of the query editor.

2. Write a condition with a variable that is marked by an ampersand (&) character.

3. Type the name of the prompt after the ampersand.
Example

For example:
SELECT DISTINCT A.AREA, A.STATE_NAME
FROM DEMO.STATEDETAILS A
WHERE (A.AREA > &S)

Working with simple prompts
You use simple prompts to limit the data retrieved from a query. Simple prompts are closely related to a query and can be saved within it.

By determining simple prompt values you can filter query results and display only those that are based on the set values. For example, you run a query against company staff data and want to display data for a specific department only. You add a simple prompt to the query and every time you run it, you are asked to input the name of the department that you are interested in.

When you create a query with a substitutional parameter and use simple prompts with default values, the values are added automatically to the query and you are not prompted to enter them manually. Query results contain data only for the department name that is specified as the simple prompt default value.

To work with simple prompts, use either the Manage prompts window or the Prompts pane on the Design tab of the Query editor.

Adding simple prompts
You can add simple prompts to a visual query, dynamart, analytical query, or compound dynamart to determine the data that is retrieved in this query so that only the specified value or ranges are used.

About this task
Use the Simple Prompt window to specify a prompt and add it to a query.

To add a simple prompt to a query:

Procedure
1. In the query editor, switch to the Design tab.
2. Use the Prompts pane on the Design tab to manage prompts.

   Note: In visual queries, the table on the Prompt Assignments pane lists the columns of the query and prompts assigned to these columns. In analytical queries, the Prompts pane lists the shared prompts that can be used for this query. For more information about shared prompts, see Working with shared prompts.

3. To create a simple prompt, click New Prompt on the Prompts pane toolbar.
4. Specify the name of the prompt in the Prompt name field of the Simple Prompt window.
5. In the Display string field, type the prompt text that is displayed when you run the query.
6. From the Input type list:
   - Select Literal for values that are enclosed within the double quotation marks if it is required by the database.
   - Select Date for values with a date part but no time part.
• Select **Time** for values with a time part but no date part.
• Select **Timestamp** for values that contain both date and time parts.
• Select **As is** for values that are passed to the database as they are entered, without any modification.
• Select **Enumeration** for values that are represented as a set of predefined constants. See the example of use at the end of the topic.

7. If you selected **Time, Timestamp, or Date** option:
   • You can specify the format string in the **Format** field. You can either choose one of standard formats or specify your own string that corresponds to Date and Time Patterns.
   • You can set the value in the Date and Time window. To open the window, click **Choose default date or time**.

8. If you need a certain value to be used automatically when you run the query, select the **Has default value** check box and specify that value in the **Default value** field.

   **Note:** If the **Has default value** check box is cleared, you are prompted to input the value when you run the query. If it is selected, the specified default value is used automatically and the prompt window is not displayed.

9. Assign the prompt to a query column by selecting one of available columns from the **Assigned column** list.

10. To edit a simple prompt, select a prompt that you want to edit from the list of available prompts on the **Prompts** pane and then click **Edit Prompt**.

11. Click **OK** to close the Simple Prompt window.

**Example**

Use the Enumeration variable type in functions that require an array as a parameter, for example, in the **in_op()** function with **getprompt()** used in the argument. The values that are entered in **getprompt()** are parsed and represented as an array for the database. Consider a query that is based on the Sample Data Source from the Samples repository:

```
SELECT SHOP_NAME, ADDRESS, CITY, STATE, ZIP, PHONE FROM DEMO.PIZZA
```

You can create a free style filter with the formula: **in_op(@[STATE], getprompt("state_arr"))**, and create the **state_arr** prompt of the enumeration type, then run the query and enter 'AK','AL' in the Prompt Variables dialog, the result set shows data only for AK, AL states.

**Saving prompt values with a query**

The Prompt values that you enter when running a query can be saved to this query.

**About this task**

The **Save prompt values** function provides you with the ability to use the last saved value as a default in the Prompt Variables window next time you run this query.

When you use a dynamart or compound dynamart with prompts, the prompt values that you enter are always saved within this dynamart.
If you decide to save prompt values, be aware that any user can see the last prompt value in the Prompt Variables window and run the query with this value as default.

To save prompt values within a query:

**Procedure**
1. With an active query in the editor, select *Query > Manage Prompts*.
2. In the Manage Prompts window, select *Save prompt values* check box. If the check box is cleared, the prompt values are not saved and you are to enter the prompt value anytime you run the query. If the check box is selected, you can view the last value in the Prompt Variables window and use it as default.

   **Note:** When you use prompts with dynamarts and compound dynamarts, the *Save prompt values* check box is always selected and cannot be cleared.
3. Click *OK* to close the window.
4. Run the query by clicking *Run Query* on the toolbar.
5. In the Prompt Variables window, input the value that you want the query to run with.
6. After the query is run, save it to a repository or file system.

**Working with prompt hierarchies**

Prompt hierarchies allow users to select one or more values from a flat or hierarchical list of options.

Prompt hierarchies are distinct DataQuant objects that are stored independent of a given query, allowing them to be used by multiple query objects. Prompt hierarchies draw their option list from a query, for example “select region from sales territories”. In addition, a prompt hierarchy can present a hierarchy of values, with unlimited levels of detail. For example, “select region, country from sales territories”. In this case, the list of options would be broken down by ‘Americas’, ‘Europe’ and ‘APAC’, with USA, Canada and Mexico listed under ‘Americas’ and so on. Users can then select values at both levels of the prompt, perhaps selecting ‘APAC’ in its entirety and ‘Americas’ but not selecting Mexico.

**Creating prompt hierarchies**

You can create a prompt hierarchy by using the Create New Prompt Hierarchy wizard.

**About this task**

To create a prompt hierarchy:

**Procedure**
1. From the main menu, select *File > New > Other* to open the "Select a wizard" page of the New wizard.
2. Expand the *DataQuant Objects* folder, select *Prompt Hierarchy*, and click *Next* to open the Create New Prompt Hierarchy wizard.
3. Specify a name for the prompt hierarchy in the *Name* field and click *Finish* to close the Create New Prompt Hierarchy wizard and open the Prompt Hierarchy editor.
4. From the **Data Source** section of the Prompt Hierarchy editor, select the source of the data that you want to use for the prompt hierarchy. The data source for a prompt hierarchy can be a visual query, analytical query, or dynamart.

   - Select **Linked**, if you want to store a link to a data source object in the prompt hierarchy that you create. Then go to step 5.
   - Select **Embedded**, if you want to save a data source within the prompt hierarchy that you create. Then go to step 6.

5. If you selected **Linked**, click the ellipsis (...) button that is next to the **Resource** field, and navigate to and select the required data source. The path to the data source that you selected is displayed in the **Resource** field.

6. If you selected **Embedded**, select one of the following options:
   - Click **Import** to navigate to and select the required data source object.
   - Click **New** to create a data source query or dynamart for the prompt hierarchy.
   - Click **Edit** to modify the query or dynamart that is already specified as the embedded data source for the prompt hierarchy.

7. In the **Hierarchy levels** section, click **Add Level** to add levels to the prompt hierarchy.

8. From the **Hierarchy levels** list, select a level that you added and specify the following properties for it:
   - In the **Level name** field, specify the name that you want to use for the level.
   - From the **Display value** list, select a column, which values are displayed when a user is prompted to enter the variable value.
   - From the **Return value** field, select a column that returns the variable value that corresponds to the display value.
     For example, the "SALES" department has a department number of "10". The department name column is set as the display value, and the department number column is set as the return value. If the user enters SALES in the prompt, 10 is returned to the query.
   - From the **Order by** list, select a column that you want to use for ordering query results.

9. To change the position of the levels in the prompt hierarchy, select a level and click **Move Level Up** or **Move Level Down**.

10. In the **Prompt hierarchy preview** section, click **Refresh** to preview the prompt hierarchy.

11. When you finished adding levels to the prompt hierarchy, save it to a repository or file.

**Adding existing prompt hierarchies**
You can add an existing prompt hierarchy that is stored in a repository or file to a visual query or dynamart.

**About this task**

To add an existing prompt hierarchy:

**Procedure**

1. In the query editor, switch to the **Design** tab.
2. On the Design tab, in the Prompts pane, click New Prompt Hierarchy (➕) to open the Prompt Hierarchy dialog.

3. In the Prompt name field, specify the name that you want to use for the prompt hierarchy.

4. In the Display string field, specify the text that you want to display when the query is run.

5. Click the ellipsis (...) button to navigate to and select the prompt hierarchy that you want to add. The path to the prompt hierarchy that you selected is displayed in the Resource field.

6. Select the type of the resource.
   - Select Embedded to store the prompt hierarchy with the query or dynamart.
   - Select Linked to store the link to the prompt hierarchy in the query or dynamart.

7. Optional: In the Prompt Level column, you can select the required level of the prompt hierarchy for each query column.

8. Click OK to close the Prompt Hierarchy dialog and return to the Design tab of the query editor. The prompt hierarchy that you added is displayed in the Prompts pane.

9. In the Prompt Assignments pane, select a column and click the down arrow in the Prompt column to select the prompt hierarchy level for the column.

10. To update data of the linked prompt hierarchy, select it in the Prompts pane and click Refresh linked prompt hierarchy.

11. To edit the level structure or data source object for the prompt hierarchy, select it in the Prompts pane and click Open in prompt hierarchy editor (➡️). The prompt hierarchy is opened in the Prompt Hierarchy editor.

### Embedding prompt hierarchies in queries

You can create an embedded prompt hierarchy to store it with a visual query or dynamart.

### Before you begin

Open a query to which you want to add an embedded prompt hierarchy on the Design tab of the query editor.

### About this task

To create a prompt hierarchy and embed it in the query:

### Procedure

1. On the Design tab, in the Prompts pane, click the down arrow and select Create Embedded Prompt Hierarchy (➕) to open the Create Embedded Prompt Hierarchy wizard.

2. In the Prompt name field, specify the name that you want to use for the prompt hierarchy.

3. In the Display string field, specify the text that you want to display when the query is run.

4. Click Finish to close the Create Embedded Prompt Hierarchy wizard and open the Prompt Hierarchy editor.
5. In the Prompt Hierarchy editor, create a prompt hierarchy and save it. For more information, see the “Creating prompt hierarchies” on page 53.

6. Return on the Design tab of the query editor. The prompt hierarchy that you created is embedded in the query and displayed in the Prompts pane.

7. In the Prompt Assignments pane, select a column to which you want to apply the prompt hierarchy and in the Prompt column, click the down arrow and select the prompt hierarchy level for the column.

8. To edit the level structure or data source object for the prompt hierarchy, select it in the Prompts pane and click Open in prompt hierarchy editor ( ). The prompt hierarchy is opened in the Prompt Hierarchy editor.

### Working with shared prompts

Use shared prompts to apply filtering to analytical queries and compound dynamarts.

#### About this task

In analytical queries, you use prompt groups that include shared child query prompts or newly added simple prompts.

To manage shared prompts in analytical queries or compound dynamarts:

#### Procedure

1. With an active query in the query editor, either select Query > Manage Prompts or open the Prompts pane on the Design tab. On the Prompts pane, you can see the child query prompts or newly added simple prompts or prompt hierarchies.

2. To add a simple prompt, perform the steps from [Adding simple prompts](#). The newly added simple prompt is automatically added to a newly created prompt group that borrows its name.

3. To move a prompt from the default group to a different group of shared prompts:
   a. From the Available prompts list, select a prompt that you want to add to a group.
   b. From the Prompt Group list, select a group to which you want to add the prompt.
   c. Click Add to Group. The selected prompt is added to the prompt group.

   **Note:** Only prompts of one type can be added to the same prompt group.

4. To remove a prompt from a prompt group:
   a. From the Prompt Group list, select the prompt that you want to remove.
   b. Click Detach Prompt. The selected prompt is detached from the prompt group.

5. To rename a prompt group:
   a. From the Prompt Group list, select the prompt that you want to rename.
   b. Click Rename Prompt Group.
   c. In the Prompt Group window, specify the name and string to display and click OK.

6. To change the order of the prompt groups in the Prompt Group list:
a. From the **Prompt Group** list, select the node of the prompt group or a prompt in the group.
b. Click **Move Prompt Group Up** or **Move Prompt Group Down**.

7. If you work in the Manage Prompts window, click **OK** to close the window.

---

### Creating OLAP queries

OLAP queries enable users to dynamically access various aspects of multidimensional data, including dimensions, hierarchies, members, titles, values, and instances, as well as rows, columns, and data points.

#### About this task

Using DataQuant for Workstation and DataQuant for WebSphere you can create an OLAP query for multidimensional data sources that is queried using MDX and accessed using XMLA.

### Retrieving cube data

OLAP queries are built based on a cube of multidimensional data. To create a new OLAP query you first must select and open a cube of data, then select the information from the cube that you want to view.

#### About this task

All the data cubes that are available for your use are listed in your workspace.

To retrieve cube data for an OLAP query:

#### Procedure

1. Open the Create New OLAP Query wizard.
2. On the **Select a data source** page, select the multidimensional data source where the cube that contains your data resides. Click **Next**.
3. The **Select a cube** page of the wizard opens. The cubes that are available on the data source are displayed in the **Cube models and cubes** list. Select a cube from the list. Click **Finish**.

   The cube information is retrieved and displayed in layout format in the editor window. In the layout format, the **Cube Structure** tree lists the dimensions and measures that are available in the cube and the **Query Structure** tree lists the dimensions and measures that are selected for the OLAP query of the cube.

   You modify the dimensions and measures that are listed in the **Query Structure** tree to build your OLAP query and obtain different views of the cube data. The structure of the OLAP query is also displayed in the **Outline** view.

4. When an OLAP query is open in the editor window, you can select one of the following tabs:
   - **Results** to open the Results page which displays the results of the current OLAP query. When you select the results tab, the OLAP query is rerun to update the results. You can use the **Outline** view and Results page to build OLAP queries dynamically.
   - **Filter** to open the Filter page where you can filter the dimensions and measures that will be retrieved from the cube.
   - **MDX** to open the MDX page which displays the MDX statements for the OLAP query. You can modify the MDX for the OLAP query.
• **SQL** to open the SQL page which displays the SQL statements for the OLAP query. This is a read only view of the SQL that was created for the OLAP query. This page is only available if the multidimensional data source uses the SQL-based API to view the SQL that has been created for the OLAP query.

• **Results XML** to open the XML page that displays the XML results for the OLAP query from the multidimensional data source. This is a read only view of the XML created for the OLAP query. This tab is available if the multidimensional data source works with XML for Analysis (XMLA).

• **Layout** to open the Layout page which displays all the dimensions and measures that are included in the cube and the dimensions and measures that are included in the current OLAP query. You use the Layout page to build an OLAP query offline.

• **Preview** to preview the OLAP query results.

### Opening existing OLAP queries

You can open existing OLAP queries that are included in your workspace.

**About this task**

To open an existing OLAP query, expand and collapse folders in your workspace until you find the OLAP query that you want to open. Queries that have been created and saved as OLAP queries are identified with a unique icon containing a small cube in front of the query name. Open the OLAP query by double-clicking the query. The query opens in an OLAP Query editor window.

### Modifying OLAP queries to obtain different views of cube data

Once you have opened a cube of multidimensional data there are several ways to query the cube and expose the information that it contains.

**Building the OLAP query offline**

Building an OLAP query offline means that you will retrieve data from your cube data source only when you have finished building or making changes to your OLAP query.

**About this task**

You use the **Layout** page to build an OLAP query offline. You will initiate running the OLAP query when you select the **Results** tab.

To build an OLAP query offline:

**Procedure**

1. Double-click a cube in your workspace or open an existing OLAP query. The OLAP query opens in the editor window.
2. An OLAP query has four tabs that run across the bottom. Select the **Layout** tab. The Layout page opens. It includes the **Cube Structure** tree and the **Query structure** tree.
3. The **Cube Structure** tree lists the dimensions and measures that are contained in the cube.
4. The **Query Structure** tree lists the dimensions and measures that have been selected for inclusion in the OLAP query. The Query Structure tree consists of three elements:
   - **Top Dimensions**: The dimension values that are included in this node are displayed across the top of the query results with data summarized.
   - **Side Dimensions**: The dimension values that are included in this node are displayed along the side of the query results (starting in the first column) as a line of summary data.
   - **Measures** display in the query results as a column of data.

5. To select a measure for inclusion in the OLAP query, click the measure in the **Cube Structure** tree. With the mouse button pressed, drag the measure to the **Measures** node of the **Query Structure** tree. Position the measure in the node in the same location as where you want the value to appear in the query results. Release the mouse button.

6. To select a dimension for inclusion in the OLAP query as a top or side dimension, click the dimension in the **Cube Structure** tree. With the mouse button pressed, drag the dimension to the **Top Dimensions** or **Side Dimensions** node of the **Query Structure** tree. Position the dimension in the node in the same order as you want its values to appear in the query results. Release the mouse button. Only dimensions can be used as top or side groups.

7. To remove a measure or dimension from the **Query Structure** tree, select it from the tree, drag it, and drop it back in the **Cube Structure** tree.

8. Click the **Results** tab to run the OLAP query. The OLAP query is run and the results are displayed on the **Results** page.

**Building an OLAP query online**

Building an OLAP query online means that you will retrieve data from your cube data source every time you make a change to an OLAP query.

**About this task**

You use the **Outline** view and the Results page to build an OLAP query online.

To dynamically build your OLAP query and see the results after each change:

**Procedure**

1. Double-click a cube in your workspace or open an existing OLAP query. The OLAP Query opens in the editor.

2. Open the **Outline** view. To open the **Outline** view, select **Window > Show views > Outline**.

3. A single structure tree is displayed in the **Outline** view or the **Layout** window. The dimensions and measures that are available in the cube are displayed under a node with the cube's name. The dimensions and measures that have been selected for the OLAP query are displayed under the **Layout** node.

4. To select a measure for inclusion in the OLAP query, click the measure in the cube's name node of the tree. With the mouse button pressed, drag the measure to the **Measures** node of the **Layout** node. Position the measure in the node in the same location as where you want the value to appear in the query results. Release the mouse button.

5. To select a dimension for inclusion in the OLAP query as a top or side dimension, click the dimension in the cube's name node of the tree. With the mouse button pressed, drag the dimension to the **Top Dimensions** or **Side Dimensions** node of the **Layout** node. Position the dimension in the node in
the same order as you want its values to appear in the query results. Release
the mouse button. Only dimensions can be used as top or side groups.

6. Each time you make a change, you will retrieve data from your cube data
source and the query results are displayed in the Results page.

Filtering cube data
Filtering enables you to limit the amount of data that is retrieved from a cube.

About this task
The Filter page lists the dimensions defined for the cube. Each dimension of a cube
has one or more attributes. By selecting a check box you can choose to include or
exclude the attribute’s or entire dimension’s data from being retrieved from a cube.

To filter the data that is retrieved from an OLAP cube:

Procedure
1. Double-click a cube in your workspace or open an existing OLAP query.
2. An OLAP query has six tabs that run across the bottom. Select the Filter tab.
The Filter page opens. It includes the Filter tree and the Filter Type fields.
3. The Filter tree lists the dimensions of the cube, the hierarchy levels for each
dimension, and the dimension values for each hierarchy level. Click the (+) to
expand the dimension.
4. To select your filtering options, select the check boxes of the dimension values
that you want included in the query. Each dimension, hierarchy level, and
dimension value has a check box. You can expand and collapse the dimensions
and hierarchies using the plus (+) and (-) boxes. You will select or clear the
check boxes to include or exclude data beginning with the lowest level of the
tree structure, the dimension values. If you select all values in a level, the
higher level of the tree structure will be selected. If no values in a level are
selected, the higher level of the tree structure will be cleared. If you select some
of the values in a level, the higher level of the tree structure will be checked
but with a grayed check box. You must select at least one value for each
hierarchical level.
5. If the data cube resides on a multidimensional data source uses the SQL-based
API, you can vary the organization of the tree depending on your selection of
filter type.
   - Select Regular Filter to display a distinct list of dimension values by simple
     hierarchical level. All possible values are listed regardless of their hierarchy
     in the dimension. For example, if you have multiple quarters within multiple
     years, each dimension value would be organized as follows:
     
     2001
     2002
     Q1
     Q2

   - Select Hierarchical Filter to organize the dimension values and display them
     under each of the parent hierarchy levels where they could possibly appear.
     This type of listing is useful when you have instances of a given value that
     can be included in multiple hierarchical levels. For example if you have
     multiple quarters within multiple years, each dimension value would be
     organized as follows:
Creating analytical queries

Analytical queries allow you to combine data from multiple queries from the same or differing data sources into one result set.

About this task

Using analytical queries, you can combine query results from multiple queries that span different data sources into one query result set. There are several principle query types that you can incorporate into an analytical query structure tree:

- **Append Query** - Append queries combine query results from two queries into one query result set. You can specify how the columns will be paired up across the two result sets.

- **Join Query** - Join queries join the result sets from two queries into one unified result set by joining one or more columns together, much like a join operation in SQL.

- **Crosstab Query** - Crosstab queries provide a cross-tabulation of one or more columns, such as a sum, average, min, max, computed over two or more tabulation columns. For example, tabulating the sum of product sales across the region and month.

- **Normalize Query** - Normalized queries "normalize" the text returned in a query result set. Normalizing query text facilitates query processing, such that executions of the same query with different parameters can be easily compared with one another and aggregated together.

- **Conditional Grouping Query** - With Conditional Grouping queries you can create aggregated summaries of result set data based on conditional expressions.

- **Column Filter Query** - With Column filter queries you can add a calculated column before or after existing columns and enter filtering functions.

Multiple queries and tables can be added into the analytical query structure tree to generate a comprehensive result set that encompass data from many different sources. The different options of each query type allow you to customize how this data is displayed.

To create an analytical query:

**Procedure**

1. Select **File > New > Other**. The New window opens.

2. Expand the **DataQuant Objects** folder, select **Analytical Query**, and click **Next**. The Create New Analytical Query wizard opens.

3. Specify a name for the analytical query in the **Name** field.
4. Click Finish. The Create New Analytical Query wizard closes and the new analytical query is displayed in the Structure query editor with a query pre-loaded in the Analytical query structure pane.

5. To add a query or table object to the analytical query structure, select the query node under which you want to add the new query, click Add, and then select the type of the object that you want to add. The new query or table object is displayed under the selected query node.

**Note:** To add a query that is currently open in a separate editor, an empty node in the analytical query structure tree, select Add From Opened, and select one of the available queries.

6. To add a table, which is saved in a data file to an analytical query, select Add Table from the query item pop-up menu and open the required file.

7. To change a query to another query type, select a table or query, click Change to and select a new query type or an open query.

**Note:** If you want replace the current query in the node with one of the queries that are opened in the editor, select the Change to Opened item.

8. To rename a table or query, select it and click Rename. The Rename Query window opens.

9. Specify the new name for the query or table in the Query name field and click OK. The Rename Query window closes and the query or table is renamed.

10. To detach a table or query, select it and click Detach. A copy of the query or table opens in a separate editor. Any changes that are made have no effect on the analytical query that it was taken from.

**Note:** You cannot detach the table that was added from a data file.

11. To open and edit a query or table in a separate editor, select it and click Open in Separate Editor. The query or table opens in a separate editor. Any changes that are made have a direct effect on the analytical query that it was taken from.

**Note:** You cannot open the table that was added from a data file in a separate editor.

**Adding an append query**

You use append queries to add selected result set columns from different queries to the final result set.

**About this task**

Append queries allow you to combine result set columns from different queries from either the same or different data sources to one result set. When you append two columns from two different queries, all of the rows from both of the selected columns are combined in one column in the final query results. The column in the First list determines the name of the final query result set column, and the rows from the column will be displayed first.

To add an append query to an analytical query:

**Procedure**

1. Create an analytical query and display it in the Structure query editor.
2. Right-click in the Analytical query structure pane and select Add Append from the pop-up menu.

3. Right-click the first node named <none> and select Add Query from the pop-up menu. The Open window opens.

4. Navigate to and select the query that you want to set as the first inner query and click Finish. The Open window closes and the query’s columns are displayed in the First list of the Item editor pane.

5. Right-click the second node named <none> and select Add Query from the pop-up menu. The Open window opens.

6. Navigate to and select the query that you want to set as the second inner query and click Finish. The Open window closes and the query’s columns are displayed in the Second list of the Item editor pane.

7. Specify the query result columns that you want to add together.
   a. Select a column from the First list.
   b. Select a column from the Second list.
   c. Click Add Column above the Result columns list. The new append condition is added to the Result columns list.
   d. Repeat this process for each append condition that you want to add.

8. If you are populating the append query with queries that contain similar names, you can attempt to automatically create the appropriate append conditions by clicking Automatch. If there are any possible automatches, they are displayed in the Result columns list.

   Note: Clicking Automatch will erase any append conditions that you previously set.

Adding a join query

You use join queries to join result set columns from different queries and tables.

About this task

Join queries allow you to join columns from different queries from either the same or different data sources to one result set. You specify any number of join conditions, and specify inner, left, right, or full joins.

To add a join query to the analytical query:

Procedure

1. Create an analytical query and display it in the Structure query editor.

2. Right-click in the Analytical query structure pane and select Add Join from the pop-up menu.

3. Right-click the first node named <none> and select Add Query from the pop-up menu. The Open window opens.

4. Navigate to and select the query that you want to draw left columns from and click Finish. The Open window closes and the query’s columns are displayed in the Left Columns list of the Item editor pane.

5. Right-click the second node named <none> and select Add Query from the pop-up menu. The Open window opens.

6. Navigate to and select the query that you want to draw right columns from and click Finish. The Open window closes and the query’s columns are displayed in the Right Columns list of the Item editor pane.
7. Specify the query result columns that you want to join together.
   a. Select the type of join that you want to add from the Join Type drop-down list.
   b. Select a column from the Left Columns list.
   c. Select a column from the Right Columns list.
   d. Click Add Join Key above the Join Columns list. The new join condition is added to the Join Columns list.
   e. Repeat this process for each join condition that you want to add.
8. To add a column to the result set from the left or right columns, select a column and click Add to Result Columns.
   To move the added result column up or down, select the column and click the To Up or To Down button above the Result Columns list.
9. To add all of the available columns, click Add All Columns above the Result Columns list.

Adding a crosstab query
You use crosstab queries to create aggregated summaries of result set data across grouped columns.

About this task
Crosstab queries are useful for isolating and displaying the aggregated values of grouped result set columns. For example, you want to know the sum of the salaries of the employees in each department of a company, grouped into three job types. With a crosstab query, you can create a result set that displays a row for each department and a column for each of the three job types. Each cell of the result set displays the sum of the salaries of the employees in given department who perform the given job type. For example, you can quickly find the sum of the salaries of all of the clerks in department number ten.

To add a crosstab query:

Procedure
1. Create an analytical query and display it in the Structure query editor.
2. Right-click in the Analytical query structure pane and select Add Crosstab from the pop-up menu.
3. Right-click the first node named <none> and select Add Query from the pop-up menu. The Open window opens.
4. Navigate to and select the query that you want to draw crosstab results from and click Finish. The Open window closes and the query is displayed in the Analytical query structure pane.
5. Click Add in the Group pane. The Select Columns window opens.
6. Select the column that you want to display in the side group and click OK. The Select Columns window closes and the column is displayed in the Columns table.
7. Click the value in the Aggregation column and select how you want to aggregate the column from the drop-down list.
8. Select the column that you want to display in the top group from the Grouping column drop-down list.
9. Select the column that will provide the descriptions for the top group columns from the Description column drop-down list.
10. Click Add in the Crosstab pane. The Select Columns window opens.

11. Select the column that will provide the values in the cells of the crosstab result set and click OK. The Select Columns window closes and the column is displayed in the Value columns table.

12. Click the value in the Aggregation column and select how you want to aggregate the values in the cells of the result set from the drop-down list.

**Adding an analytical query**

You can add an analytical query to the Analytical query structure tree at any point.

**About this task**

Previously created analytical queries can be added to the analytical query structure tree.

**Procedure**

1. Create an analytical query and display it in the Structure query editor.
2. Right-click in the Analytical query structure pane and select Add Analytical Query from the context menu.
3. Navigate to and select the analytical query that you want to add and click Finish. The Open window closes and the analytical query is displayed in the Analytical query structure pane.

**Adding a normalize query**

Normalization of query result data allows you to collapse multiple columns into two columns within a new result set.

**About this task**

Each distinct original column value is placed within a new, single data column in the new result set, one row per column value. A second column is used to indicate which original column value a given row includes.

**Table 7. Original result set data**

<table>
<thead>
<tr>
<th>Region</th>
<th>Revenue</th>
<th>Units Sold</th>
<th>Net Income</th>
</tr>
</thead>
<tbody>
<tr>
<td>North America</td>
<td>250</td>
<td>452</td>
<td>87</td>
</tr>
<tr>
<td>Asia Pacific</td>
<td>290</td>
<td>538</td>
<td>92</td>
</tr>
<tr>
<td>Europe</td>
<td>320</td>
<td>675</td>
<td>120</td>
</tr>
</tbody>
</table>

Normalization allows us to transform the data into the following two forms:

**Table 8. Normalized query results**

<table>
<thead>
<tr>
<th>Region</th>
<th>KPI</th>
<th>KPI Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>North America</td>
<td>Revenue</td>
<td>250</td>
</tr>
<tr>
<td>North America</td>
<td>Units Sold</td>
<td>452</td>
</tr>
<tr>
<td>North America</td>
<td>Net Income</td>
<td>87</td>
</tr>
<tr>
<td>Asia Pacific</td>
<td>Revenue</td>
<td>290</td>
</tr>
<tr>
<td>Asia Pacific</td>
<td>Units Sold</td>
<td>538</td>
</tr>
<tr>
<td>Asia Pacific</td>
<td>Net Income</td>
<td>92</td>
</tr>
</tbody>
</table>
Table 8. Normalized query results (continued)

<table>
<thead>
<tr>
<th>Region</th>
<th>KPI</th>
<th>KPI Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Europe</td>
<td>Revenue</td>
<td>320</td>
</tr>
<tr>
<td>Europe</td>
<td>Units Sold</td>
<td>675</td>
</tr>
<tr>
<td>Europe</td>
<td>Net Income</td>
<td>120</td>
</tr>
</tbody>
</table>

Table 9. Alternate normalized query results

<table>
<thead>
<tr>
<th>Region</th>
<th>KPI</th>
<th>KPI Value</th>
<th>Units Sold</th>
</tr>
</thead>
<tbody>
<tr>
<td>North America</td>
<td>Revenue</td>
<td>250</td>
<td>452</td>
</tr>
<tr>
<td>North America</td>
<td>Net Income</td>
<td>87</td>
<td>452</td>
</tr>
<tr>
<td>Asia Pacific</td>
<td>Revenue</td>
<td>290</td>
<td>538</td>
</tr>
<tr>
<td>Asia Pacific</td>
<td>Net Income</td>
<td>92</td>
<td>538</td>
</tr>
<tr>
<td>Europe</td>
<td>Revenue</td>
<td>320</td>
<td>675</td>
</tr>
<tr>
<td>Europe</td>
<td>Net Income</td>
<td>120</td>
<td>675</td>
</tr>
</tbody>
</table>

In Table 8 on page 65, the first transformation, all three numeric columns have been normalized into a column pair. In Table 9, Revenue and Net Income have been normalized and Units Sold has been retained as an independent column. Note that the names of both the column that carries the value (KPI in this sample) and the column that indicates the original column (KPI Value) are arbitrary and can be defined by the user.

Normalized queries are often used to reverse an aggregation, rotating the pivot of the table and displaying individual rows for aggregated data. However, it is important to note that normalization can be applied to any result set data, not merely aggregated values.

Queries containing aggregated data can be normalized, in effect reversing the cross-tabulation and enabling the query results data to be displayed in a normalized format. For example, the Furniture Sales table is made up of the following columns Order Date, Category and Gross Profit, to display the gross profit of sales by the order date. The table is then crosstabbed to display a Gross Profit Column for each Category and an Order Date Column.

The crosstabbed query results appear as shown in Table 10.

Table 10. Crosstabbed query results for furniture sales

<table>
<thead>
<tr>
<th>Order Date</th>
<th>Gross Profit (CHAIRS)</th>
<th>Gross Profit (SOFAS)</th>
<th>Gross Profit (MISC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>19 November 2008</td>
<td>160</td>
<td>452</td>
<td>87</td>
</tr>
<tr>
<td>23 November 2008</td>
<td>0</td>
<td>680</td>
<td>181</td>
</tr>
<tr>
<td>26 November 2008</td>
<td>120</td>
<td>642</td>
<td>0</td>
</tr>
</tbody>
</table>

Normalizing the query reverses the cross-tabulation to produce query results that will look like the original query results before the cross-tabulation was applied. The Normalized query results appear as shown in Table 11.

Table 11. Normalized query results for furniture sales

<table>
<thead>
<tr>
<th>Order Date</th>
<th>Category</th>
<th>Gross Profit</th>
</tr>
</thead>
<tbody>
<tr>
<td>19 November 2008</td>
<td>CHAIRS</td>
<td>160</td>
</tr>
<tr>
<td>19 November 2008</td>
<td>MISC</td>
<td>87</td>
</tr>
</tbody>
</table>
Table 11. Normalized query results for furniture sales (continued)

<table>
<thead>
<tr>
<th>Order Date</th>
<th>Category</th>
<th>Gross Profit</th>
</tr>
</thead>
<tbody>
<tr>
<td>19 November 2008</td>
<td>SOFAS</td>
<td>452</td>
</tr>
<tr>
<td>23 November 2008</td>
<td>MISC</td>
<td>181</td>
</tr>
<tr>
<td>23 November 2008</td>
<td>SOFAS</td>
<td>680</td>
</tr>
<tr>
<td>26 November 2008</td>
<td>CHAIRS</td>
<td>120</td>
</tr>
<tr>
<td>26 November 2008</td>
<td>SOFAS</td>
<td>642</td>
</tr>
</tbody>
</table>

To add a normalize query:

**Procedure**

1. Create an analytical query and display it in the *Structure* query editor.
2. Right-click in the *Analytical query structure* pane and select *Add Normalize* from the pop-up menu.
3. Right-click the first node named `<none>` and select *Add Query* from the pop-up menu. The Open window opens.
4. Navigate to and select the crosstabbed or aggregated query that you want to normalize and click *Finish*. The Open window closes.
5. In the *Dimensions* section, click *Add* above the *No Transform* list box, select the columns that you want to include without alterations and click *OK*. These columns will be displayed without transformation and will contain repeated values for each row of normalized columns. In the examples shown in this topic, the *Order Date* column is the *No Transform* column.
6. Click *Add* above the *Columns* list box. The Dimension Column window opens.
7. Specify the name and type of the column in the *Name* field and *Type* drop-down list. This is the new column that will display the names of the normalized columns. In Table 11 on page 66, this is the *Category* column.
8. Specify the names of the normalized columns that will be displayed in the *Constants* table and click *OK*. In Table 11 on page 66, these are the different product type columns, such as *CHAIRS*, *MISC*, and *SOFAS*. These are labels for the normalized column names and as such can be anything. For example, if you wanted to make the query results more readable, you could specify lowercase names like chairs, misc, and sofas.
9. Click *Add* above the *Values* list box. The Value Column window opens.
10. Specify the name and type of the column in the *Name* field and *Type* drop-down list and click *OK*. This is the new column that will display the values of the normalized columns. In Table 11 on page 66, this is the *Gross Profit* column.
11. In the *Normalize Columns* section, click *Add*, select the columns that you want to normalize and click *OK*. These are the columns that you want to move down into the specified dimension columns. In Table 10 on page 66, these are the *Gross Profit (CHAIRS)*, *Gross Profit (SOFAS)* and *Gross Profit (MISC)* columns.
12. Select the value of the normalized column from the *Value* drop-down list.
13. Select the category of the normalized column from the *Category* drop-down list.
14. Select the *Suppress Zeroes* or *Suppress Nulls* check boxes to exclude rows from the normalized result set that contain values of zero or no values at all.
15. Once you have finished adding normalize columns, click the Run Query toolbar button. The query is run and the normalized results are displayed in the Results editor.

Adding a conditional grouping
Conditional groupings allow you to create aggregated summaries of result set data based on conditional expressions.

About this task
A conditional grouping lets you create filters that control what values from a given value column are included in the aggregated summary. For example, you have a table that contains staff data for your company. You create a query that displays the sum of the salaries of all of the employees in a given department across each job type. Now, you want to only display the sum of the salaries of employees that have been working for five or more years. You can use a conditional grouping to create a filter that only includes in the aggregation salary values from rows with a value of 5 or more in the YEARS column.

<table>
<thead>
<tr>
<th>Table 12. Original data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Department</td>
</tr>
<tr>
<td>SALES</td>
</tr>
<tr>
<td>SALES</td>
</tr>
<tr>
<td>SUPPORT</td>
</tr>
<tr>
<td>R&amp;D</td>
</tr>
<tr>
<td>R&amp;D</td>
</tr>
</tbody>
</table>

Using a conditional grouping, the result set can be transformed into the following:

<table>
<thead>
<tr>
<th>Table 13. Conditionally grouped data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Department</td>
</tr>
<tr>
<td>SALES</td>
</tr>
<tr>
<td>SUPPORT</td>
</tr>
<tr>
<td>R&amp;D</td>
</tr>
</tbody>
</table>

This conditional grouping was done by carrying out the following steps:
1. Adding Department and Name as grouped columns. Department was added as a GROUP and Name was added as a COUNT. The Name column allows us to include a column showing total number of employees in a department, providing a contrast with the number used to perform the salary summation.
2. A filter was defined to select employees with 5 or more years service.
3. Two conditional columns were added, both driven by the defined filter. Total Salary performs a sum of the Salary column and Matched Employees performs a count of the Name column.
4. Finally, the Name column in the result set was edited to display Total Employees via the Design tab (right-click the column and select Change Column Heading from the pop-up menu).

To add a conditional grouping:
Procedure

1. Create an analytical query and display it in the Structure query editor.
2. Right-click in the Analytical query structure pane and select Add Conditional Grouping from the pop-up menu.
3. Right-click the first node named <none> and select Add Query from the pop-up menu. The Open window opens.
4. Navigate to and select the query that you want to draw columns from and click Finish. The Open window closes.
5. In the Group section, click Add. The Select Columns window opens.
6. Hold down the Ctrl key, select the column names that you want to group aggregated results by, and click OK. The Select Columns window closes and the group columns are displayed in the Columns table. Aggregated values will be broken down for each row of a given group column.
7. Specify how a group column will be aggregated by clicking in the Aggregation column and selecting an aggregation method from the drop-down list.
9. Specify a name for the filter in the Filter name field. It is a good idea to give the filter a name that reflects its function, for example, Older Group for the filter that will display employees that have worked more than five years.
10. Select a filter building method and click Next. The next page of the wizard depends on your previous selection.
11. Build your filter using the table of options or the expression designer and click Finish. The Filter Wizard closes and the filter that you specified is displayed in the Filters table.
12. In the Columns table, select the column that you want to filter from the Source column drop-down list. This column will provide the values that will be filtered and aggregated across the group columns.
13. Select the filter that you will use with this column from the Filter drop-down list. This list is populated with all of the filters that you built in the Filters table.
14. Select how the values will be aggregated from the Aggregation drop-down list.
15. Specify a name for the column of aggregated values in the Column name field.
16. Once you have finished adding all of the filtered value columns that you want to display, click the Run Query toolbar button. The query is run and the query results are displayed in the Results editor.

Results

You will notice that there is a column for every group and value column that you selected. The value column headers display the column names that you specified. The contents of the value columns represent all of the values that are included in the conditional filters that you set and are aggregated according to the methods that you selected.

For more information about building filters, see:

- “Building a compound condition filter” on page 107
- “Building a free style condition filter” on page 108
Adding a column filter query

You can use column filter queries to add calculated columns and filters to the source query without modifying the source query itself. You can also choose to exclude columns from the result query.

About this task

For example, let's say you have a query that displays staff data for your company and that this query uses a calculated column to enumerate the rows. When you apply a filter to this query, some rows are excluded from the result set. The remaining rows (the rows not excluded) maintain their original position, meaning their position is preserved and is not recalculated based on the rows excluded. You can use a column filter query to add another calculated column that displays the row numbers of the result query and not the source.

To add a column filter query:

Procedure

1. Create an analytical query and display it in the Structure query editor.
2. Right-click the Analytical query structure pane, and then select Add Column Filter.
3. Right-click the <none> node, and then select Add Query.
4. In the Open window navigate to and select the source query that you want to use. The source query’s columns are displayed in the Result Columns list.
5. Specify the query result columns that you want to be displayed by clicking Move to the Left or Move to the Right. You can also click Move All to the Left and Move All to the Right to move all the columns simultaneously between the Source Columns and the Result Columns lists.
6. To apply sorting conditions to the columns that you included to the Result Columns list, specify values in the Sort Order and Key Sequence columns.
   - Sort order specifies whether the column is used for sorting data in the query result set. Select the Ascending, Descending, or None sort order from the list for the appropriate column.
   - Key sequence specifies the sequence in which the sort order conditions are applied to the columns. Select the sequence number from the list for the columns with sort order conditions (the column that has the key sequence number ’1’ is sorted first and so on).
7. Click the Design tab. On the Design page you can manage prompts, calculated columns, and filters.
8. In the Prompts pane, you can add and edit prompts.
9. Use the Calculated Column window to add calculated columns. Click in the Dynamart pane, and then select Add Calculated Column.
10. Use the Dynamart Filter wizard to create conditional expressions that control the contents of query results. In the Transformations pane right-click the Filter node and select Edit Filter.

Example

The following table shows a query with calculated column which returns row numbers:
If you apply a filter to this query, some rows will be dropped from the result set. But the calculated column will return the same row numbers as in the source query, as shown in the following table:

<table>
<thead>
<tr>
<th>Deptname</th>
<th>Deptnumb</th>
<th>Division</th>
<th>Location</th>
<th>Manager</th>
<th>Row</th>
</tr>
</thead>
<tbody>
<tr>
<td>HEAD OFFICE</td>
<td>10</td>
<td>CORPORATE</td>
<td>NEW YORK</td>
<td>160</td>
<td>0</td>
</tr>
<tr>
<td>NEW ENGLAND</td>
<td>15</td>
<td>EASTER</td>
<td>BOSTON</td>
<td>50</td>
<td>1</td>
</tr>
<tr>
<td>MID ATLANTIC</td>
<td>20</td>
<td>EASTER</td>
<td>WASHINGTON</td>
<td>10</td>
<td>2</td>
</tr>
<tr>
<td>SOUTH ATLANTIC</td>
<td>38</td>
<td>EASTER</td>
<td>ATLANTA</td>
<td>30</td>
<td>3</td>
</tr>
<tr>
<td>GREAT LAKE</td>
<td>43</td>
<td>MIDWEST</td>
<td>CHICAGO</td>
<td>100</td>
<td>4</td>
</tr>
<tr>
<td>PLAINS</td>
<td>51</td>
<td>MIDWEST</td>
<td>DALLAS</td>
<td>140</td>
<td>5</td>
</tr>
</tbody>
</table>

You can use column filter query to add another calculated column that will show row numbers of the result query, as shown in the following table:

<table>
<thead>
<tr>
<th>Deptname</th>
<th>Deptnumb</th>
<th>Division</th>
<th>Location</th>
<th>Manager</th>
<th>Row</th>
<th>Row2</th>
</tr>
</thead>
<tbody>
<tr>
<td>HEAD OFFICE</td>
<td>10</td>
<td>CORPORATE</td>
<td>NEW YORK</td>
<td>160</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>GREAT LAKE</td>
<td>43</td>
<td>MIDWEST</td>
<td>CHICAGO</td>
<td>100</td>
<td>4</td>
<td>1</td>
</tr>
<tr>
<td>PLAINS</td>
<td>51</td>
<td>MIDWEST</td>
<td>DALLAS</td>
<td>140</td>
<td>5</td>
<td>2</td>
</tr>
</tbody>
</table>

For more information about adding calculated columns and building filters, see:

- Adding calculated columns to the query results
- Filtering query results

**Working with Cloudant queries**

In DataQuant for Workstation and DataQuant for WebSphere you can create Cloudant queries to retrieve data from Cloudant databases.

**About this task**

Cloudant query is an HTTP request (usually with some set of parameters) to a Cloudant database that is run against an index.

In DataQuant you can create the following types of Cloudant queries:

- Queries that are based on a secondary or search index.

  Secondary index (or view) is the index that is defined by a JavaScript “map” function, which retrieves keys and values from the JSON documents based on the contents of each document, and an optional "reduce" function, which aggregates the data.

  Search index is the index that is defined by a JavaScript function, which determines the fields in JSON documents that can be queried.
• Queries that include a selector object. These queries are called Cloudant native queries.

Selector is a JSON structure that defines the list of fields and the corresponding values for the fields that you want to extract from JSON documents.

The following topics contain information about Cloudant functionality in DataQuant:

Creating Cloudant queries
You create a Cloudant query to retrieve data from the index that is built based on the content of a Cloudant database.

About this task

To create a Cloudant query:

Procedure
1. From the main menu select, File > New > Other to open the "Select a wizard" page of the New wizard.
2. Expand the DataQuant Objects folder, select Cloudant Query, and click Next to open the Create Cloudant Query wizard.
3. In the Name field, enter the name that you want to use for a query.
4. From the Data sources field, select the Cloudant data source that defines the connection to the database from which you want to retrieve the results.
5. From the Type field, select the type of the query that you want to create.
   • Select Index Query to create a query by using a secondary or search index that is defined in the Cloudant database. Then go to step 6.
   • Select Native Query to create a query with a selector. Then go to step 7.
6. If you selected the Index Query option, in the Cloudant data source tree, expand the Secondary Indexes or Search Indexes folder, and select the index that you want to use for the query.
7. Click Finish to close the Create Cloudant Query wizard and open the query in the editor.
   If you create a Cloudant query based on a secondary or search index, the index that you selected is built and opened as a metadata table on the Build tab of the Cloudant Query editor.
   If you create a Cloudant native query, it is opened on the Build tab of the Cloudant Native Query editor.
8. On the Build tab, specify the appropriate settings and query parameters that depend on the type of the query that you create. For more information, see the following topics:
   • "Creating queries in Cloudant Query editor" on page 73
   • "Creating queries in Cloudant Native Query editor" on page 74
9. When you finished building the query, click the Run Query toolbar button to run the query and retrieve its results. The query results are displayed as a grid on the Results tab and as a JSON tree on the JSON tab.

Tip: Use the Output view to see the HTTP request of the Cloudant query that you run. To display an HTTP request in the Output view, ensure that the Show query text check box is selected on the Output page of the Preferences window.
10. Switch to the **Preview** tab, to see the formatted read-only view of query results.

11. Save the Cloudant query that you created to a repository or file.

**Results**

**Note:** To work with Cloudant query results, for example, to apply filtering, add a calculated column, create a chart, and so on, add a Cloudant query to an analytical query and manage the results of an analytical query. For more information about analytical queries, see the “Creating analytical queries” on page 61 section.

**Creating queries in Cloudant Query editor**

In the Cloudant Query editor, you specify settings and parameters for Cloudant queries to retrieve specific data to query results.

**Before you begin**

By using the Create Cloudant Query wizard, create and open a Cloudant query that uses a secondary or search index on the **Build** tab of the Cloudant Query editor.

**About this task**

To specify the options for a Cloudant query, do the following steps:

**Procedure**

1. To change the index for the query, click the **Change Index** button. In the Change Index dialog, from the **Secondary Indexes** or **Search Indexes** folders, select the index that you want to query, and click **OK**.

   **Tip:** To see the functions that are defined for building the index, switch to the **Index** tab.

2. On the **Build** tab, in the **Field definitions** section, you can change the metadata that defines the result set.
   - To change the position of the columns in the query results, select a column and click the **Move Up** or **Move Down** navigation buttons.
   - To add a column to the result set, click **Add Column** and specify a column name and a JSON field, which will supply column values.
   - You can edit the data type of the query columns in the **Type** column.

3. Select the **Ignore conversion errors** check box to run the query and display query results even if the data cannot be properly converted to the data type that is specified for the column.

   **Tip:** If the data type conversion errors occur, the appropriate messages are displayed in the **Output** view.

4. Select the **Ignore system documents** check box not to query the documents with ids that begin with "/design/".

5. In the **Structure parameters** section, specify the parameters that define the organization of the data in the results.

   **Tip:** When you change structure parameters, the data in the **Field definitions** section is updated automatically.

6. In the **Filter parameters** section, specify the parameters that filter the result set using multiple conditions.
7. Select the check boxes in the Share column for the filter parameters that you want to use as prompts when a Cloudant query is run inside an analytical query.

8. When you finished building the query, you can run it to retrieve query results and save it to a repository or file.

Creating queries in Cloudant Native Query editor
In the Cloudant Native Query editor, you specify a selector object and settings for Cloudant native queries to retrieve specific data to query results.

Before you begin
By using the Create Cloudant Query wizard, create and open a Cloudant native query on the Build tab of the Cloudant Native Query editor.

About this task
To specify the options for a Cloudant native query:

Procedure
1. In the Selector field, specify the JSON body to extract the documents with the required fields and values from a Cloudant database.
2. In the Field definitions sections, click the Refresh Field Definitions button to build the metadata table.

   Note: The metadata table cannot be built, if the index for the selector that you specified is not defined. For more information about creating indexes for Cloudant native queries, see the "Managing indexes for Cloudant native queries" topic.
3. In the Field definitions section, you can change the metadata that defines the result set.
   • To change the position of the columns in the query results, select a column and click the Move Up or Move Down navigation buttons.
   • To add a column to the result set, click Add Column and specify a column name and a JSON field, which will supply column values.
   • You can edit the data type of the query columns in the Type column.
4. Select the Ignore conversion errors check box to run the query and display query results even if the data cannot be properly converted to the data type that is specified for the column.

   Tip: If the data type conversion errors occur, the appropriate messages are displayed in the Output view.
5. When you finished building the query, you can run it to retrieve query results and save it to a repository or file.

Managing indexes for Cloudant native queries:
You can add, edit, or remove indexes for Cloudant native queries.

Before you begin
Open a Cloudant native query on the Build tab of the Cloudant Native Query editor.
To define an index:

Procedure

1. From the main menu, select **Query > Manage Indexes** to open the Manage Indexes dialog.
2. Click **Add Index** to open the Manage Index dialog.
3. In the Manage Index dialog, specify the name that you want to use for the index.
4. Click **Add Field** to specify the name of the field that you want to include to the index from database documents. You can specify multiple field names.
5. When you specified all fields that you want to index, click **OK**. The index that you created is added to the **Index names** list of the Manage Indexes dialog. The fields that you specified for including in the index are displayed in the **Fields** section.
6. To edit an index, select the required index from the **Index names** list and click **Edit Index**.
7. To delete an index, select the index from the **Index names** list and click **Remove Index**.

    **Note:** The Manage Indexes dialog contains the service index with the name ",_all_docs" that cannot be edited or removed.
8. Click **OK** to close the Manage Indexes dialog and return to the Cloudant Native Query editor.

Results

The indexes that you added for Cloudant native queries are included in the **Query Indexes** folder under the appropriate design document node in the Repository Explorer view.

**Working with Cloudant indexes**

In DataQuant, you can use secondary and search indexes that are defined in a Cloudant database, add new indexes, edit, and remove them.

About this task

**Note:** You must have writing permissions to the database to create, edit, or remove indexes.

See the following topics for more information about Cloudant indexes:

**Opening Cloudant indexes**

You can open secondary and search indexes that are defined in a Cloudant database to edit or run them as Cloudant queries.

About this task

To open an index:

Procedure

1. In the Repository Explorer view, expand the **Cloudant Data Sources** node.
2. Navigate to and expand the Cloudant data source that contains the index that you want to open.

3. Expand the node with the name that corresponds to the name of the design document that contains the index that you want to open.

4. Expand the Secondary Indexes or Search Indexes folder to display the list of available indexes.

5. To rename an index, right-click the required index and select Rename from the menu.

6. To edit an index, right-click the required index and select Open With > Cloudant Secondary Index Editor or Open With > Cloudant Search Index Editor depending on the type of the index that you want to edit. The secondary index is opened in the Cloudant Secondary Index editor. The search index is opened in the Cloudant Search Index editor. In the editors, you can edit the functions that are defined for building the index.

7. To open an index in the Cloudant Query editor, right-click the required index and select Open With > Cloudant Query Editor from the menu. The index is built and opened as a metadata table on the Build tab of the Cloudant Query editor.

8. Click the Run Query toolbar button to display the indexed data as a grid on the Results tab and as a JSON tree on the JSON tab.

9. On the Build tab, you can modify a metadata table and specify parameters to display specific data in the results.

10. To save changes that you made on the Build tab, save the index as a Cloudant query to a repository or file.

Creating secondary indexes
You can add new secondary indexes by defining the "map" and "reduce" functions to retrieve the specific data from Cloudant databases.

About this task

To create a secondary index:

Procedure
1. From the main menu, select File > New > Other to open the "Select a wizard" page of the New wizard.
2. Expand the Repository node, select Cloudant Index, and click Next to open the Create Cloudant Index wizard.
3. From the Data source list, select a Cloudant data source to which you want to add a new index.
4. From the Design document list, select the design document in which you want to include the new index.
5. In the Index name field, enter the name that you want to use for the index.
6. From the Index type list, select Secondary Index and click Finish to create the index and open it in the Cloudant Secondary Index editor.
7. In the Map function field, specify the "map" function that retrieves keys and values from the documents based on the content of each document.
8. From the Reduce list, select a "reduce" function that you want to use to aggregate the data.
   - Select None, if you do not want to apply any "reduce" function.
   - Select _sum to return the sum of all values for a key.
   - Select _count to return the count of the rows for a key.
   - Select _stats to return the JSON structure that contains sum, count, min, max, and sum squared.
   - Select Custom Reduce Function to specify your own "reduce" function in the Custom reduce function field.

9. Click to save the secondary index that you created to the database.

Results

The secondary index that you created is added to the Secondary Indexes folder under the appropriate design document node in the Repository Explorer view.

Creating search indexes

You can add new search indexes by defining the search index functions to retrieve the specific data from Cloudant databases.

About this task

To create a search index:

Procedure

1. From the main menu, select File > New > Other to open the "Select a wizard" page of the New wizard.
2. Expand the Repository node, select Cloudant Index, and click Next to open the Create Cloudant Index wizard.
3. From the Data source list, select a Cloudant data source to which you want to add a new index.
4. From the Design document list, select the design document in which you want to include the new index.
5. In the Index name field, enter the name that you want to use for the index.
6. From the Index type list, select Search Index and click Finish to create the index and open it in the Cloudant Search Index editor.
7. In the Search index function field, specify the function to determine the fields in documents that can be queried.
8. In the Analyzer section, specify the settings to define the rules for recognizing the terms within text.
   - Select Single, if you want to apply the same analyzer for all document fields. Then from the Type list, select the analyzer type.
   - Select Multiple, if you want to configure multiple analyzers for different fields. Then click Add Field to specify the name of the field in the Field Name column and select the analyzer type that is applied to this field from the Analyzer column.
9. Click to save the search index that you created to the database.
Results

The search index that you created is added to the Search Indexes folder under the appropriate design document node in the Repository Explorer view.

Profiling queries

You use profiling to analyze the complexity of the queries.

Before you begin

You must have a query to analyze open.

About this task

You can profile the following query types containing only SELECT statements:

- queries
- visual queries
- analytical queries

You can also profile Cloudant queries.

To profile a query:

Procedure

1. Open the required query in the Query Editor.
2. Click the Profile Query toolbar button. The Query Profiler view opens. It displays the time spent on running the query and fetching its results if there are any.
3. Click the SQL Text toolbar button in the Query Profiler view to see the SQL text of the last profiled query. The SQL text is displayed in the Query Profiler window.
4. For Cloudant queries, click the Cloudant Query URL toolbar button in the Query Profiler view to see the detailed URL of the profiled Cloudant query. The URL is displayed in the Query Profiler window.

Working with database tables

You can open any database table that is accessible to you in your workspace using a default editor, the Table Editor, the Table Viewer or the Query Editor.

About this task

To open a database table:

Procedure

1. Right-click the table in any view and select Open With.
2. Click one of the following items:
   - Table Viewer to display the contents of the table in the Query Results view.
   - When a table is opened with Table Viewer, a query is created, run and the results are returned.
• **Query Editor** to display the table in the **Query Builder** editor. When the table is opened with the **Query Editor**, a query is created and displayed in the **Query Builder** editor. The query is not run and results are not returned. Users can modify the query using the **Query Builder** editor.

• **Table Editor** to display the table in edit mode.

The default view for the Table Editor is the Results view, as indicated by the Results tab in the editing area.

**Note:** Permission to use the Table Editor is set by the administrator. Click **View > Resource Limits** and select the **Options** tab to verify whether you have permission to use the Table Editor. If you have permission to edit tables, the **Enable table editing** property is set to Yes.

You can use the Table Editor to search for, add, edit, and delete the data that is stored in the tables without writing SQL statements.

**Using the Results menu item to work with database tables**

The Results menu item is activated when viewing a table in the Results view of the Table Editor.

The Results menu provides access to most of the functionality in the Table Editor.

To access and use Result menu options:

a. With the table opened in the Table Editor, select the Results tab.

b. Click Results from the menu list to display available options.

The following table lists the Results menu options and describes the purpose of each option.

<table>
<thead>
<tr>
<th>Option</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>Retrieve All</td>
<td>Select this option to retrieve all rows from the table.</td>
</tr>
<tr>
<td>Find</td>
<td>Select this option to search the table for specific data. Enter the search criteria in the Find dialog and click Find next.</td>
</tr>
<tr>
<td>Insert Row</td>
<td>Select this option to insert a table row.</td>
</tr>
<tr>
<td>Delete Row</td>
<td>Select this option to remove a table row. The row that you select for deletion is marked with an asterisk and is removed if you commit the change to the table.</td>
</tr>
<tr>
<td>Edit Row</td>
<td>Select this option to edit row data.</td>
</tr>
<tr>
<td>Commit</td>
<td>Select this option to manually commit the changes you have made to the table</td>
</tr>
<tr>
<td>Rollback</td>
<td>Select this option to undo changes that you have made to the table</td>
</tr>
<tr>
<td>Immediate Commit</td>
<td>Select this option to enable the immediate commit function, which results in changes to the table being committed immediately.</td>
</tr>
</tbody>
</table>

**Editing content directly in the table**

You can edit table data directly in the Table Editor in the following ways in:

- Placing your cursor in the cell containing the information you want to edit, right-clicking and selecting **Edit** from the pop-up menu.
In the **Cell value** window, enter your changes and click **OK**. Click **OK** again to confirm that the updates will committed to the database.

Click **Cancel** if you want to roll back the updates.

- Placing your cursor in the cell and pressing **Enter** or by double-clicking in the table cell.

### Changing the table structure

You change the structure of a table (for example, the number and data type of columns) directly in the **Table Editor** in the following ways:

- Deleting a row or multiple rows:
  a. Click the row number of the row that you want to delete. To select multiple rows, press the **Shift** key while clicking the table rows.
  b. Right-click and select **Delete Row** from the pop-up menu.
  c. If you are sure that you want to delete the row from the table, click **Yes**. Click **No** if you decide not to delete the row from the table.

- Editing a table row:
  a. Click the row number of the row that you want to edit.
  b. Right-click and select **Edit Row** from the pop-up menu.
    The row and its content are displayed in an Edit Row window.
  c. Make changes to data in the **Value** of the Edit Row window and click **OK**.
  d. Click **OK** to apply your updates and commit changes to the database, or click **Cancel** to roll back the changes.

- Inserting a row:
  a. Select **Insert Row** from the **Results** menu.

    If the option **Immediate Commit** on the menu is selected, the changes that you make to the table are applied to the database *as you make them* and click **OK**.

    If the option **Immediate Commit** is cleared then the changes that you make are stored locally and are applied all at once when you select the Commit option, or they are disregarded all at once if select the Roll back option.

    Otherwise, when you close the editor object you will get a message to Commit or Roll back all pending changes.

    The inserted row will be the last row of the result set after the commit is done.

**Note:** If you select a row before you click **Insert Row** then the Insert dialog will be pre-populated with the content of the row so that you can use it as a template for a new row.

To specify how to display the nonnumeric table data, place your cursor in the cell containing text string for which you want to set the display format. Right-click, select **Edit** from the pop-up menu and then click **Advanced**.

From the **Format mode** list, select **Hexadecimal** to display the data in hexadecimal format or **Binary** to display the data in binary format. The Zoom window displays samples of the data in the format that you have chosen.

You can use the Prompted Query view of the Table Editor to set the sort and row conditions in the table. For information about setting sort and row conditions, see [Specifying row conditions for query results](#).
• **Visual Query Editor** to display the table in the **Query Builder** editor. When the table is opened with the **Visual Query Editor**, a query is created and displayed in the **Query Builder** editor. The query is not run and results are not returned. Users can modify the query using the **Query Builder** editor.

• **Default Editor** to display the table using the editor that was last used.

### Specifying sort conditions for tables

You can specify the sort conditions that determine the order that the rows are displayed in tables.

#### About this task

Use the **Sort conditions** pane of the **Prompted** menu to set sort conditions for tables. The **Prompted** menu of the **Table** editor is similar to the **Prompted Query** editor, except that only the **Sort conditions** and **Row conditions** panes are available.

#### Procedure

1. Right-click a table in the **Workspaces** view and select **Open With > Table Editor**. The table opens in the editor window.

2. Click the **Prompted** tab. The **Prompted** menu opens with only the **Sort conditions** and **Row conditions** panes available.

3. Click the **Add Sort Condition** button in the **Sort conditions** toolbar. The Sort Conditions window opens.

4. Specify the columns that will be used to sort the query results in one of the following ways:
   - Select one or more columns from the **Columns in the result set** list. The **Columns in the result set** lists the columns that are included in the table. You can select one or more columns if you are adding a sort condition. You can select only one column if you are changing a sort condition.
   - Specify a conditional expression in the **Or, enter a condition here** field. Click the ellipsis (...) to open an expression builder that will help you build the conditional expression. The expression builder offers a palette of common elements used to create SQL expressions, such as column names, constants, functions, and operators. When you click the expression builder buttons, templates for expression elements are inserted into the expression fields.

5. Specify the sort direction. Select **Ascending** to specify that the query results will be sorted in ascending (lowest-to-highest) order. Select **Descending** to specify that the query results will be sorted in descending (highest-to-lowest) order. The first column is used for the primary sort; subsequent columns will sort within the first.

6. Click **Add** to add the sort condition to the table. If you are changing a sort condition, click **Change**. Repeat this process for each column that you want included in the sort conditions. When you have finished selecting the columns that will be included in the sort conditions, click Close. The Sort Conditions window closes.

### Specifying row conditions for tables

If you are using DataQuant for Workstation, you can specify row conditions to limit the rows that are displayed in the table. Without row conditions, all existing rows will be displayed.
About this task

You use the Row conditions pane of the Prompted menu to specify the row conditions for tables. The Prompted menu of the Table editor is similar to the Prompted Query editor, except that only the Sort conditions and Row conditions panes are available.

Procedure

1. Right-click a table in the Workspaces view and select Open With > Table Editor. The table opens in the editor window.
2. Click the Prompted tab. The Prompted menu opens with only the Sort conditions and Row conditions panes available.
3. Click Add Row Condition in the Row conditions toolbar. The Row Conditions window opens.
4. If you are specifying more than one condition that must be met for a row to be included in the table, you must indicate how each condition will be connected using the Connector radio buttons. If you select And, then the current condition that you are specifying and the previous condition (listed in the Row conditions pane of the Prompted menu) must be met in order for the row to be included. If you select the Or, then the current condition or the previous condition can be met in order for the row to be included. When you connect multiple conditions using both And and Or, then the rules of the data source determine how the conditions are processed.
5. Select a column from the Select a column from the list list box to examine for inclusion in the table. The Select a column from the list list box lists all the columns that will be included in the table. To specify a conditional expression that will determine the column, enter an expression in the Or enter an expression here field. Click the ellipsis (...) to open an expression builder that will help you build the conditional expression.
6. Specify the operator for the row condition expression using the Operator field. To use the operator as is, select Is. To change the operator to its negative opposite, click the Is Not. For example, to specify "Is Not Equal To" select Is Not and select the Equal To operator in the list. You cannot use the Starting with, Ending with, or Containing operators if the left and right sides of the row condition expression have numeric data types. You cannot use any of the spatial predicates unless the left and right sides have spatial data types.

Note: If IBM DB2 Spatial Extender is installed on the database, you are able to work with spatial data types: points, lines, and polygons. But you cannot use any of the spatial predicates unless the left and right sides have spatial data types.
7. Specify the condition for which to check the rows in the Right Side area.
   a. You can use the expression builder button to help you build the conditions.
   b. To add row conditions from a file of any format with the values that are separated by commas, click Import Conditions From File.
   c. To specify row conditions from a set of query results, click Import Conditions via Executing SQL Query, and enter the SQL text of a query.
   d. You can use the Remove Condition or Remove All Conditions buttons to remove one or all row conditions.
8. Click Add. The row condition is added and displayed in the Row conditions pane of the Prompted menu. Repeat this process for each condition that you want to add. When you have finished specifying each condition, click Close. The Row Conditions window closes. You are returned to the Prompted menu.
Editing database tables using DB2 special registers

You can change data in database tables using DB2 special registers. A special register is a storage area that is defined for an application process by DB2 and is used to store information for the reference. A reference to a special register is a reference to a value provided by the current server. The list of available special registers depends on the version of the database.

Before you begin

Open the database table that you want to edit on the Results tab of the Table Editor.

About this task

To edit a database table using special registers:

Procedure
1. Enter a special register to a table cell, which value you want to edit.
2. Commit the changes to replace the special registers that you entered by the values of the special registers. See the “Working with database tables” on page 78 topic for the detailed information on editing, inserting data, and committing changes to the tables.

Note: If you use the special register to change the column value in one of the duplicate rows of the table, the column values are updated in all duplicate rows.
The database casts the data type of the special register value to the column data type.

Managing data in Cloudant databases

In DataQuant, you can view the JSON documents that a Cloudant database contains, edit or remove them, and add new documents.

About this task

Note: You must have writing permissions to the database to add, edit, or remove documents.

Working with Cloudant documents

You can display documents from the Cloudant database to which you are connected to view document structure and content.

About this task

To display Cloudant documents:

Procedure
1. In the Repository Explorer view, expand the Cloudant Data Sources node.
2. Navigate to and expand the Cloudant data source that defines the connection to the database with the documents that you want to display.
3. Double-click the All Documents node to open the All Cloudant Documents editor and display all documents that the database contains.
4. To display specific documents from the database, click **Show Query Options** and specify the parameters for retrieving the documents.

5. Click **Refresh Documents** to update the list of the documents based on the parameters that you specified.

   **Tip:** To quickly find the required document by its ID number, enter the ID in the **Type document ID** filter field and click **Find.**

**Adding documents to Cloudant databases**

You can add new documents to Cloudant databases.

**About this task**

To add a Cloudant document:

**Procedure**

1. From the main menu, select **File > New > Other** to open the "Select a wizard" page of the New wizard.

2. Expand the **DataQuant objects** folder, select **Cloudant Document**, and click **Next** to open the Create Cloudant Document wizard.

3. From the **Data source** list, select the Cloudant data source that defines the connection to the database to which you want to add a document.

4. Optionally enter the ID number that you want to use for the document in the **ID** field.

   **Tip:** If you do not specify the document ID, it is generated automatically.

5. Click **Finish**, to create a document and open it in the Cloudant Document editor.

6. In the **Document content** field, specify the content of the document in the JSON format.

7. In the **Service Fields** section, you can view the ID number that is assigned to the document in the **_Id** field and the revision number that is added by the server when you add or modify the document in the **_Rev** field.

8. Click **to save the document that you created to the database.**

9. After you saved the document, you can add BLOB objects to it. To add BLOB objects to the document, in the **Attachments** section, click the **Add** button to navigate to the required file and attach it to the document. The name of the attached files, their types, and sizes are displayed in the Attachments table.

   **Tip:** To download the file that is attached to the document from the database to your local machine, select it in the Attachments table and click **Download**. To remove the attached file from the document, select it and click **Remove.**

**Editing documents in Cloudant databases**

You can edit documents that a Cloudant database contains.

**About this task**

To edit a Cloudant document:
Procedure

1. In the Repository Explorer view, expand the Cloudant Data Sources node.
2. Navigate to and expand the Cloudant data source that contains the document that you want to edit.
3. Double-click the All Documents node to open the All Cloudant Documents editor and display all documents that the database contains.
4. Move to the document that you want to modify and click the Edit button to open the document in the Cloudant Document editor.
   
   Tip: To quickly find the required document by its ID number, enter the ID in the Type document ID filter field and click Find.
5. In the Cloudant Document editor, modify the document.
6. When you finished modifying the document, click to save the changes to the document.
7. To remove the document from the database, in the All Cloudant Documents editor, select the required document and click Remove.
Chapter 4. Selecting a new data source

You use the Set Data Source dialog to select a data source against which a query object will be run.

About this task

All query objects are associated with a default data source. When query objects are run, they are automatically run against the default data source. The default data source for a query object is the last used data source before the query was saved or in the case of new queries that have not yet been saved, the default is the data source last used in the DataQuant session.

To select the data source where you want to run the current query object:

Procedure

1. Open the Set Data Source window by selecting the Set Data Source command from one of the following menus:
   - Query
   - Form
   - Procedure
2. Select the data source where you want to run the query object from the Data source list. The Data source list includes all the data sources that are available for your use based on your user ID.
3. In the User name field, type the user ID that you want DataQuant to use to connect to the new data source.
4. In the Password field, specify the password associated with the user ID that you typed in the User name field.
5. To save the password that you have specified for this user ID and use it every time to connect to this data source, select the Remember this password check box.
6. Click OK. The object is run against the data source that you have specified in the Data Source field.
Chapter 5. Setting fonts for SQL statements

If you are using DataQuant for Workstation, you can set unique font attributes for the SQL statement text.

About this task

To set unique font attributes:

Procedure
1. Display SQL statements in the workstation query editor.
2. Open a the Font window one of the following ways:
   • Select Query > Set Font.
   • Select Form > Set Font.
   • Select Procedure > Set Font.
3. Select the font for the SQL statements that are displayed in the query window from the list of available fonts in the Font field.
4. In the Font style field, specify the style of the font.
5. In the Size field specify the size of the font.
6. Click OK to set the new font settings that you have specified. The Font window closes.
Chapter 6. Specifying substitution variables

Substitution variables are used to enter changing values to a SQL query at run time.

About this task

This feature enables you to substitute a part of an SQL statement and make it more generic. Substitution variables are active only while the object (query, form, or procedure) is running. As a result, only one object can access the substitution variable. The variable will not exist after the object is executed.

A substitution variable is special text in a query that begins with an ampersand character (&). A substitution variable can contain up to 18 alphabetic, numeric or special characters.

A substitution variable can appear anywhere in a query. The value of the substitution variable can be anything used in a query (except a comment). For example, you can use a substitution variable in place of a column name, search condition, subquery, or any specific value.

In the following example, you will be prompted for a customer number each time you run the following query:
SELECT ORDERNO, SALESREPNO, PRODNO, QUANTITY, &CUSTNO AS CUSTOMER# FROM Q.SALES

When you run the query and supply customer number at the prompt, the query will retrieve only those records that are associated with the specified customer number. Later you can launch the query and provide a different customer instead of writing a separate query.

To use a substitution variable:

Procedure
1. Open a query.
2. Type this SQL statement: SELECT * FROM Q.STAFF WHERE DEPT = &MIN_DEPT
3. Run the query. The Enter Substitution Variable Values window opens.
4. Type 50 in the Value field of the window.
5. Click OK. The query will run with the value 50 for DEPT.

What to do next

Try experimenting with substitution variables by replacing values in the SELECT and FROM clauses. See what results your queries return.
Chapter 7. Working with procedures

A procedure is a set of commands that enable you to run queries, print reports, import and export data, as well as perform other functions.

Use the Procedure window to create, open, display, and run procedures. A procedure is a set of DataQuant procedure commands that enable you to run queries, print reports, import and export data, as well as perform other functions. Procedures can be saved in a repository, in a QMF catalog, or to a file. All commands issued through procedures are governed by your resource limits. Procedures can contain:

- any DataQuant procedure command
- comment lines
- blank lines
- RUN commands that run other procedures or queries
- substitution variables

Note:
- DataQuant supports legacy job files. You must convert the legacy job files before they can be run. For Windows operating systems, you can schedule converted job files to run using the Windows Scheduler. You can also run converted job files from the DataQuant for Workstation command line.
- DataQuant supports procedures with logic (REXX procedures). Procedures with logic include REXX language directives. Users must have OBJECTREXX installed to run procedures with logic.

Creating a new procedure

You can use the Procedure window to create new procedures.

About this task

To create a procedure:

Procedure

1. From the User perspective, select File > New > Procedure. The Create Procedure wizard opens. You can also select the New Procedure toolbar button.
2. Specify a name for the new procedure in the Name field.
3. Select from the list of available data sources in the Data source field where the new procedure will reside if saved. If you leave this field blank, the new procedure will be saved on the current data source.
4. Click Finish. The new procedure object is opened in the editor view of the User perspective.
5. Enter the commands that will be included in the procedure in the Procedure editor. If you are creating a procedure with logic using REXX language directives, you must type a REXX comment line as the first line of the procedure. The REXX comment line should appear as /*REXX*/. In addition, if you are including any DataQuant procedure commands in the procedure with logic, the DataQuant procedure commands must be enclosed in single quotes.
6. Save, run or print the procedure. After running a procedure, results are returned in the available editor window. If you are running a procedure with logic, communication with the procedure is through the REXX Console view.

7. You can schedule a procedure to run at another time using the DataQuant for WebSphere task scheduler or your local scheduler.

Converting legacy job definition files

Job definition files created using QMF for Windows must first be migrated to a new format before they can be run using DataQuant for Workstation.

About this task

To migrate legacy job definition files to a new format:

Procedure

1. Select File > Import. The Import wizard opens.
3. Specify the directory where the legacy job definition files (.jdf) can be found in the job files directory. You can use the Browse button to search for the directory.
4. Select the job files that you want to migrate to the new format from the Job files list. Each .jdf file that resides in the directory is listed in the Job files list.
5. Click Next. The .jdf files will be prepared for migration. The Server mapping page of the wizard opens.
6. Before the migration to the new format can be completed you must identify the data sources that will be used in the jobs. In QMF for Windows, data sources were referred to as servers and were defined in server definition files (SDF). Each server that is used in the legacy job file is listed in the QMF Server cell of the Server mapping table as it appeared in the SDF. These servers must be configured in the repository as data sources. This is usually done by the DataQuant for Workstation administrator. In the Repository Data Source cell of the Server mapping table specify the data source name for the server as it was configured in the repository.
7. To specify the name and password that will be used to login to the data source, select the data source and click Set User Information. The Set User Information window opens. Specify the name and password. Click OK. The information is listed in the Repository Data Source cell of the Server mapping table.
8. Specify the directory where the new job definition files (.jdfx) will be saved in the Output directory field. You can use the Browse button to search for the directory.
9. After you complete the process of migrating the job file to the new format you can run the job in one of the following ways:
   • Schedule the DataQuant for Workstation job file (.jdfx) as a task using your operating system’s scheduler. For more information on scheduling tasks, see the documentation for your scheduler.
   • Run the job file (.jdfx) using the DataQuant for Workstation command line using the following as an example:
     C:\Program Files\IBM\IBM DataQuant\dataquant.exe
     C:\temp\jobs\test_job.jdfx/batch
Creating JavaScript procedures

You can use JavaScript syntax to create more complex and not linear procedures.

About this task

To create a JavaScript procedure:

Procedure

1. In the Procedure editor, type /*JavaScript*/. The current procedure is processed as a JavaScript one.
2. Type a script for the JavaScript procedure in the Procedure editor. Use the proc object and its functions.

Note: JavaScript procedures support the syntax of JavaScript modules.

Restriction: JavaScript procedures work with calculated columns only in result sets of visual queries.
3. Save or run the created procedure.

Creating procedures as flowcharts

You can use the enhanced procedure editor to create linear or complex JavaScript procedures as flowcharts.

About this task

To add a command to a flowchart procedure:

Procedure

1. Open the procedure editor and click the Design tab. The Procedure Designer opens.
2. To add a procedure command, double-click the block on the Design tab. You can also drag the command that you want to add from the Palette view to the block.
3. In the Add Command wizard from the Command list, select a command, which you want to add to the procedure.
4. For some commands select an object, to which you want to apply the command, from the Objects list.
5. If the command with the specified object requires parameters, click Next, and specify them on the second page of the Add Command wizard:
   a. In the [command_name] Command area, enter required values to corresponding fields. You can click Browse to search for the object that you want to add or specify the directories that you want to use in commands.

   Tip: Use the Browse button to add double quotation marks to the object name automatically. If you enter an object name manually, do not forget to add double quotation marks to it.
   b. For some commands, select one of the following options:
      - Current object to use the object that is opened in the application.
      - Object to use the object that is saved in the repository or QMF catalog.
      - JavaScript variable to use a previously created JavaScript variable. See an example of a JavaScript variable and its use in a JavaScript procedure:
Example
JavaScript variable:
```javascript
var tableName = proc.prompt("Enter a table name", "");
```
JavaScript procedure:
```javascript
/*JavaScript*/
var tableName = proc.prompt("Enter a table name", "");
proc.exec('DISPLAY TABLE ' + tableName);
```
c. Specify the command parameters in the **Parameters** area.
6. Click **Finish** to close the wizard and add the command to the procedure. Follow these steps to add each command to the procedure.

**Tip:** Use **Comment** option to add comments to the commands in the flowchart. They are visible only in the **Procedure Designer**.

**Editing flowchart procedures**
You can edit existing procedure commands in the **Procedure Designer**.

**About this task**
To edit a flowchart procedure:

**Procedure**
1. Open the procedure that you want to edit on the **Design** tab.
2. Double-click the command that you want to edit. The Edit Command wizard opens.
3. Add the changes that you want to apply to the command.

**Tip:** To replace any command with a new one, use the Edit Command wizard.
4. To add a command before or after an existing command, drag it from the Palette to the command before or after the existing command.
   a. Specify the command position on the first page of the Add Command wizard. Click **Next**
   b. On the second page of the wizard, specify the command and its parameters. Click **Finish**.
5. You can also modify the appearance of the procedure flowchart by doing the following actions:
   - To add all commands one after another by default, right-click the **Design** tab and select **Auto Layout**.
   - To add commands in the custom order, right-click the **Design** tab and clear **Auto Layout**.
   - To edit the location of the procedure block, drag it to a new position or right-click the block and select the **Up** and **Down** items.
   - To edit the alignment of the procedure blocks, right-click the required block, select **Align**, and specify the alignment that you want to apply to the current block.
   - To match the procedure blocks by height or width, right-click the block, select **Match**, and specify the option that you want to apply to the current block.
Adding JavaScript conditions to procedures

You can add JavaScript conditions to procedures to make them complex and not linear.

About this task

To add a JavaScript condition in the Procedure Designer:

Procedure

1. Open the procedure to that you want to add a condition on the Design tab.
2. From the Palette, select the Condition option and drag it to the empty block of the procedure chart.
3. In the JavaScript Condition window, type the condition that you want to include in the procedure.
4. Click OK to add the condition.

Results

• Under the Yes arrow, specify the command or condition that you want to apply if the first condition is met.
• Under the No arrow, specify the command or condition that you want to apply if the first condition is not met.
Chapter 8. Managing query results

With query results in the available editor window you can perform several formatting tasks.

Formatting the query results

You can customize how the query results will appear in the editor window.

About this task

You can specify different fonts, colors and text alignments for each column header and each cell of the column. You can also specify that column and cell formatting be applied based on the results of a conditional expression.

Using the Properties view

From the Properties view you can quickly apply many formatting options to the query results.

About this task

To apply formatting options:

Procedure

1. By default the Properties view opens with the User perspective. At any time you can open the Properties view by selecting Window > Show View > Properties.
2. Select a cell or a column in the query results. The properties for the cell or column are listed in the Properties view.
3. To apply formatting, modify one of the properties that are listed in the Properties view.

Using the Layout Properties dialog box

On the Layout Properties dialog box you can set all the options that are available for formatting query results.

About this task

The Layout Properties dialog box uses a tree structure to represent the columns in the query results. Depending on what you have selected in the query results, the Layout Properties tree displays branches for all columns in the query results or for a single column in the query results. You can apply formatting options to entire columns, individual cells, column headers, and summary cells. The options are divided into the following groups:

- General
- Font
- Format
- Conditional

To apply formatting to the query results:
Procedure

1. Open the Layout Properties dialog box in one of the following ways:
   - Right-click in a column's header cell and select **Font** or **Format** from the pop-up menu.
   - Select **Results > Font**.
   - Select **Results > Format**.

2. The **Layout** tree opens differently depending on what you have selected from the query results. If you have not selected a specific column, then the tree displays all the columns in the query results. If you have selected a column, the tree displays the branch associated with that column. With only one or more columns selected and displayed, the **Show all columns** check box is available. Select this check box to display all the query results columns in the **Layout** tree. Clear the check box to display only the selected columns.

3. Click the plus sign to the left of the column name to expand the branch. For each column in the query results, the **Layout** tree has three branches. They are named **Header**, **Detail**, and **Grand Total**. The names of the branches are the same for each column. If expandable, there is plus sign to the left of each branch. Click the plus sign to expand the branch.

4. The level of the branch that you select in the **Layout** tree determines what formatting you can perform on the column of data. As you select each branch, the Layout Properties dialog changes. The formatting fields that can be used on the selected branch are made available.

5. To specify the formatting options that will apply to the entire column, select a column name branch. The **General** page opens within the Layout Properties dialog box. From this page you can specify a header name, column width, and summary row height.

6. To specify the font and formatting options that will be used for the column's header in the query results, select the **Header** branch for the column. The **Font** and **Format** pages open within the Layout Properties dialog box.

7. You can specify default and conditional formatting for the detail and summary cells of the column. Default formatting applies to all the cells in the column except for those that meet specific conditions. Conditional formatting applies to any cells in the column that meet specific conditions.

8. To specify the default font and formatting options that will be used for each cell in the column, expand the **Detail** branch for the column, then select the **Default** branch. The **Font** and **Format** pages open within the Layout Properties dialog box. Select the **Font** tab to specify the formatting options that will apply to the font that is used to display the data in the column's detail cell. Select the **Format** tab to specify how the data will be formatted in the detail cell of the column.

9. To specify conditional formatting options for a column's detail cells, expand the **Detail** branch for the column, then select the **Default** branch or an existing condition branch. The fields used for specifying conditional formatting become available. Specify the condition name and expression. You can add one or more conditions for the detail cells.

10. To specify the default font and formatting options that will be used for a column's summary line, expand the **Grand Total** branch for the column. The **Font** and **Format** pages open within the Layout Properties dialog box. Select the **Font** tab to specify the formatting options that will apply to the font that is used to display the data in the column's summary line cell. Select the **Format** tab to specify how the data will be formatted in the summary line cell of the column.
11. To specify conditional formatting options for a column's summary cell, expand the **Grand Total** branch for the column, then select the **Default** branch or an existing condition branch. The fields used for specifying conditional formatting become available. Specify the condition name and expression. You can add one or more conditions for the summary line cell.

12. Click **Apply**. The **General**, **Font**, or **Format** options that you have specified are saved.

13. You can repeat this process to set up unique formatting options for each column in the query results. You can also duplicate the options that you have specified for one column and apply them to other columns using either of the following buttons:

- **Apply to all columns**: Use this button to apply the **Font** or **Format** options that you have specified to all the columns that are currently selected from the query results. For example, if you specify special font options for the **Detail Default** branch of a column, and select the **Apply to all columns** button, the font options that you have specified will be applied to the **Detail Default** branch of each column that is currently selected and displayed in the **Layout Properties** tree.

- **Apply to all levels**: Use this button to apply the **Font** or **Format** options that you have specified for the current level of the column to all levels of the column. For example, if you specify special font options for the **Detail Default** branch of a column and select the **Apply to all levels** button, the font options that you have specified will be applied to the **Grand Total Default** branch for the column.

14. Click **OK**. The Layout Properties dialog box closes. The format information that you have specified is saved and applied to the columns in the query results.

**Specifying general column formatting options**

On the **General** page of the Layout Properties dialog you change the header text for a column, the column width, or the height of a row. In addition, you can see the grouping or aggregation value that has been applied to a column and the name of the column.

**About this task**

General formatting options apply to all cells in a query results column including the header cell, each detail cell, and the column's summary line cell.

To specify the **General** formatting options:

**Procedure**

1. Open the Layout Properties window and click a **column name** branch. The **General** page opens within the Layout Properties window.
2. The **Heading** field displays the name of the column as it appears in the query results. You can change the header text by typing a new name in this field.
3. The **Grouping and aggregation** field displays the aggregation codes that have been specified for the column. If none have been specified, this field is not blank. This field is informational and cannot be modified.
4. The **Width** field specifies the width of the column in pixels. You can change the width by typing a new number in the field.
5. The **Summary row height** field specifies the height of the summary column (if aggregation is selected) in pixels. You can change the height by typing a new number in the field.
6. The **Source** field identifies the column that is currently selected from the **Layout Properties** tree. If the selected column is a calculated column, the conditional expression that was used to create the results in the column is displayed. This is an informational field and cannot be modified.

7. Click **Apply** to save the general formatting options that you have specified.

**Specifying font formatting options**

On the **Font** page of the Layout Properties window, you can specify unique font and color attributes for each column's header cell, for each detail cell in every column, for each break point summary cell (if break columns have been specified), and for the column's final summary cell.

**About this task**

To specify the font formatting options:

**Procedure**

1. Open the Layout Properties window. Click the **Font** tab.

2. To specify the font attributes for the header cell of the column, select the **Header** branch under the column name.
   
   Specify the font attributes that you want for the column's header cell in the **Font**, **Font style**, **Size**, **Foreground**, and **Background** fields.

3. To specify the font attributes for the detail cells of the column, select the **Detail** branch under the column name.
   
   a. From the **Detail** branch, select the **Default** branch to specify the font attributes that will be used for all detail cells in the column.
   
   b. Select a **Condition** branch to specify the font attributes that will be used for a detail cell that meets a specific condition.

   c. You can create multiple conditional statements for the detail cells in the column.

   Once you have selected the **Default** or a **Condition** branch, specify the font attributes that you want applied to the qualifying detailed cells in the **Font**, **Font Style**, **Size**, **Foreground color**, and **Background color** fields.

4. To specify the font attributes for the summary cell of the column, select the **Grand Total** branch under the column name.

   Summary cells are not always visible in the query results column. The summary cell is visible only if you have specified a grouping and aggregation code for the column.

   a. From the **Grand Total** branch, select the **Default** branch, to specify the font attributes that will be used for the summary cell.

   b. Select a **Condition** branch to specify the font attributes that will be used for the summary cell if the value of the cell meets a specific condition.

   You can create multiple conditional statements for the summary cell in the column. Once you have selected the **Default** or a **Condition** branch, specify the font attributes that you want in the **Font**, **Font Style**, **Size**, **Foreground color**, and **Background color** fields.

5. Click **Apply** to save the font attributes that you have specified.

The formatting information that you have specified for the column is applied in the query results.
What to do next

You can continue formatting the query results columns or click OK to close the Layout Properties window.

Font fields:

Descriptions for each of the font attributes that can be applied to a query result column's header, detail and summary cells.

Font  The name of the font, such as Arial or Times Roman.
Font style  The style of the font, such as bold or italic.
Size  The point size of the font.
Script  The type of script being used, for example Western.
Foreground color  The color of the text, such as dark blue.
Background color  The color the text is against, such as a gray background.
Sample  A rendering of the current formatting.
Set as default  Sets the current font attributes as the default for subsequent query results. The default will remain in effect until you choose a new font or change the default.
Reset to default  Resets the font formatting attributes to the default setting.

Specifying formatting options

On the Format page of the Layout Properties window, you can specify unique attributes that will control how the data is positioned and displayed (i.e. number of decimal positions) in the query results columns.

About this task

You can specify different format attributes for each column's header cell, for each detail cell in the column, for each break point summary cell (if break columns have been specified), and for the column's final summary cell. The available Format options vary depending on whether you selected a column with character, numeric, date, or time data.

To specify data format options:

Procedure

1.  Open the Layout Properties window. Click the Format tab.
2.  To specify format attributes for the header cell of the column, select the Header branch under the column name. Specify the format information that you want for the column's header cell in the Format, Horizontal Alignment, Vertical Alignment, and Options fields.
3.  To specify format attributes for the detail cells of the column, select the Detail branch under the column name. From the Detail branch, select the Default branch, to specify the formatting that will be used for all detail cells in the
column. Select a **Condition** branch to specify the formatting that will be used for a detail cell that meets a specific condition. You can create multiple conditional statements for the detail cells in the column. Once you have selected the **Default** or a **Condition** branch, specify the formatting that you want in the **Format, Horizontal Alignment, Vertical Alignment**, and **Options** fields.

4. To specify the formatting for the summary cell of the column, select the **Grand Total** branch under the column name. Summary cells are not always visible in the query results column. The summary cell is visible only if you have specified a grouping and aggregation code for the column. From the **Grand Total** branch, select the **Default** branch to specify the formatting that will be used for the summary cell. Select a **Condition** branch to specify the formatting that will be used for the summary cell if the value of the cell meets a specific condition. You can create multiple conditional statements for the summary cell in the column. Once you have selected the **Default** or a **Condition** branch, specify the formatting that you want in the **Format, Horizontal Alignment, Vertical Alignment**, and **Options** fields.

5. Click **Apply** to save the format attributes that you have specified. The formatting information that you have specified for the column is applied in the query results. You can continue formatting the query results columns or click **OK** to close the Layout Properties window.

**Format fields:**

Descriptions for each of the format attributes that can be applied to a query result column’s header, detail and summary cells.

**Format**

- **As is** can be applied to numeric, character, date and time data.
- **Currency** can be applied only to numeric data.
- **Decimal** can be applied only to numeric data.
- **Scientific** can be applied only to numeric data.
- **Percent** can be applied only to numeric data.
- **User Currency** can be applied only to numeric data.
- **Hexadecimal** can be applied only to character data.
- **Binary** can be applied only to character data.
- **YYYYxMMxDD** can be applied only to date data.
- **MMxDDxYYYY** can be applied only to date data.
- **DDxDDxYYYY** can be applied only to date data.
- **YYxDDD** can be applied only to date data.
- **MMxDDxYY** can be applied only to date data.
- **DDxDDMxxYY** can be applied only to date data.
- **HHxMMxSS** can be applied to time data.
- **Custom** can be applied to different data types.

**Sample**

Shows a snapshot of how the data will be formatted.

**Horizontal alignment**

From the drop-down list, select how to align data horizontally in the column. Options are:

- **Left**
- **Right**
Vertical alignment
From the drop-down list, select how to align data vertically in the column. Options are:
- Top
- Center
- Bottom

Options
- **Thousands separator** can be applied to numeric data. Select this check box to place a thousands separator in numeric values (the symbol is taken from operating system settings).
- **Negative numbers** can be applied to numeric data. Select the format for a negative number from the list of available options.
- **Decimal places** can be applied to numeric data. Specify the number of decimal places that will be included in the numeric value.
- **Wrap text** can be applied to character data. Select the check box to wrap the contents of the cell to the next line within the cell.
- **Separator symbol** can be applied to date and time data. From the list of available formats, select one that will be used to format the date or time data. For example, select a slash (/) for date data to format the data as: 03/25/02. Select a symbol to format time data, such as a colon (:). An example is: 12:03:02.
- **12 hour mode** can be applied to time data. Select the check box to display time in a 12-hour interval, such as 03:12:30 PM or 10:05:07 AM. If this check box is cleared, time is displayed in 24-hour format (such as 15:12:06).

Set as default
Sets the current format attributes as the default for subsequent query results. The current settings will remain in effect until you change the default.

Reset to default
Resets the format attributes to the default setting.

The following formats are available for date:
- YYYYxMMxDD
- MMxDDxYYYY
- DDxMMxYYYY
- YYxMMxDD
- MMxDDxYY
- DDxMMxYY

The following formats are available for time:
- HHxMMxSS
- HHxMM

The following formats are available for timestamp:
- YYYYxMMxDD HHxMMxSS
- DDxMMxYYYY HHxMMxSS
- YYYYxMMxDD HHxMM
Specifying conditional formatting options

Conditional formatting options can be set for an entire column of data in the query results or for each detail, summary and grand total cell in a column. Based on the evaluation of a conditional expression, each column and cell in the query results can have a unique set of formatting attributes.

About this task

For example, within a column, you could specify that if a cell's content equals 0, then display the cell with a red background and if the value is 100, display the cell with a blue background.

To set conditional formatting:

Procedure

1. Open the Layout Properties window. You can specify conditional formatting from either the Font page or the Format page depending on what results you want to apply to the data in the cell or column.

2. Apply conditional formatting.
   a. Click the Detail branch to apply conditional formatting to the detail cells of the column.
   b. Click the column name Total branch to apply conditional formatting to the summary cell of the column.
   c. Click the Grand Total branch to apply conditional formatting to the grand total cell of the column.

3. With the appropriate branch selected, click the Add icon. The control fields Condition name and Condition Expression become available.

4. Provide a unique name to each conditional expression that you create by typing the name of the expression in the Condition name field. A branch is automatically created in the tree using the condition name. Each condition must have an expression.

5. Type the expression in the Condition Expression field. You must follow the rules for building conditional formatting expressions. The expression language supports constant expressions, expressions defined as variables sourced on other columns in the result set (1, 2, for example), numeric operators (such as +, -, *, /), character operators (+, CONCAT), numeric and character functions (such as MIN, MAX, AVG, SQRT, CONCAT, SUBSTR), and a set of logical and relational operators (such as , =, !=).

6. After you have created the expression, select the Font or the Format page of the Layout Properties window to specify the formatting that will be applied to any cell in the column that meets the conditions set in the expression.

Filtering query results

You can use the Dynamart Filter Wizard to create conditional expressions that control the contents of query results.
About this task

Filters allow you to restrict the returned rows and values of a query result set to your specifications. Any filter can contain any number of expressions that give you complete control over the content of the grid. To filter query results.

Procedure

1. Open the query result set that you want to filter in the Results editor.
2. Select Results > Filter Results. The Dynamart Filter Wizard opens.
3. Select the Enable filtering check box. From here, you can choose to build your filter one of two ways; either use a table of options to assist you in building your filter, or use the expression designer to build a free style filter.
4. Select a filter building method and click Next. The next page of the wizard depends on your previous selection.
5. Build your filter using the table of options or the expression designer and click Finish. The Dynamart Filter Wizard closes and the filters that you specified are applied to the query results.

Building a compound condition filter

A compound condition filter allows you to filter query results using multiple conditions.

About this task

Use a compound condition filter to set multiple conditions to filter your query results. You can add more than one condition from a table of options in the Dynamart Filter Wizard. To build a compound condition filter:

Procedure

1. Open the Dynamart Filter Wizard, select the Enable filtering check box, select Compound condition filter and click Next. The Edit dynamart filter page opens.
2. Select the column that you want to filter from the Column drop-down list.
3. Select the operator of the filter from the Operator drop-down list. Valid options include:
   - **Not Less Than** - All values that are equal to or greater than the specified value are included.
   - **Greater Than** - All values that are greater than the specified value are included.
   - **Not Greater Than** - All values that are equal to or less than the specified value are included.
   - **Less Than** - All values that are less than the specified value are included.
   - **Equal To** - Only values that are equal to the specified value are included.
   - **Not Equal To** - All values that are not equal to the specified value are included.
   - **Like** - All values that match the specified pattern are included. The pattern can contain the following special characters:
     - % matches zero (0) or more characters.
     - _ matches only one (1) character.
     - ? matches only one (1) character.
   - **Unlike** - All values that are not like the specified value are included.
• **Is Null** - Only null values are included.
• **Is Not Null** - All of the values that are not null are included.
• **In Range** - All of the values that fall within the two specified values, along with the values themselves, are included.
• **Outside Range** - All of the values that fall outside of the two values, along with the values themselves, are included.
• **Between** - All of the values that fall within the two specified values are included.
• **Not Between** - All of the values that fall outside of the two specified values are included.

4. Select the type of value that you want to specify from the **Type1** drop-down list. Valid options include:
   - **Constant** - The specified value is a set constant, such as a numeral or character.
   - **Column** - The specified value is another column in the result set.
   - **Prompt** - The specified value is a prompt hierarchy that is associated with the query.

5. Specify the value of the expression in the **Value1** column.

6. If you selected an applicable operator type, specify a type and value in the **Type2** and **Value2** columns.

7. If you have multiple expressions listed, select the condition from which the expressions are run from the **Condition** drop-down list.

8. Click **Move Up** or **Move Down** to change the order in which the different expressions are run.

9. Click **Down Level** or **Up Level** to change the priority level of each expression. An expression with a lower level number has more priority than an expression with a higher level number.

10. To remove an expression, select it and click **Remove**.

11. Once you are done building and arranging your expressions, click **Finish**. The Dynamart Filter Wizard closes and the query results are filtered.

### Building a free style condition filter

A free style condition filter allows you to filter query results using the expression designer.

### About this task

One way to filter your query results is to build a set of expressions from the expression designer. This method is recommended for anyone who is familiar with the DataQuant expression designer syntax. To build a free style condition filter:

### Procedure

1. Open the Dynamart Filter Wizard, select the **Enable filtering** check box, select **Free style condition filter** and click **Next**. The Edit dynamart filter expression page opens.

2. Using proper expression syntax, write the expression that you to use to filter the results in the **Expression** field.

3. Use the function categories on the right to help you construct your filter expressions.
Note: You can use your own functions to construct the filter expression. For more information, see Adding custom functions.

4. Once you have finished building your filter expressions, click Finish. The Dynamart Filter Wizard closes and the query results are filtered.

Working with calculated columns

Calculated columns are columns of data that you can add to the query result set.

Having access to the appropriate syntax reference information and function reference information before using the calculated column feature will help you to build expressions more quickly.

For information about syntax, see Syntax for calculated column expressions in the reference section of the User help.

For a list of all available functions, including the functions for calculated columns, see Functions for calculated columns in the reference section of the User help.

Adding calculated columns to a query result set

You use the Calculated Columns window to add a calculated column to the query result set. You can add a calculated column before or after existing columns.

About this task

The content of an added column is calculated by using an expression that you define. The expression can use data from other columns and constants. The expression used to calculate the value of the added column is saved with the query. The calculated column will be included in the query result set each time the query is run.

To add a calculated column to the query result set, perform the following steps:

Procedure

1. Open the Calculated Column window in one of the following ways:
   - If you are using DataQuant for Workstation, on the Results page of the editor, right-click a column’s header in the query results and select either Add Calculated Column Before . . . or Add Calculated Column After . . . from the pop-up menu.
   - With query results in the active editor, select Results > Add Calculated Column.

2. In the Calculated Column window, specify the name of the calculated column as it should appear in the query results in the Name field.

3. Select the type of the calculated column from the Type drop-down menu. The type determines the format of the calculated column data. Valid choices are:
   - Integer: Data is expressed as whole numbers. For example, "5" or "100".
   - Decimal: Data is expressed as a real number. For example, "10.5" or "0.25".
   - Character: Data is expressed only as specific text strings. For example, "John Smith", or "10 Main Street".
   - Date: Data is expressed in a date format. For example, "12/03/2000" or "1/2/09".
   - Boolean: Data is expressed as a Boolean value. For example, "true or false", or "yes or no".
• **Categorization**: Data is represented by category names that are defined for a range of data values. For example, “small, average and large” can be used to categorize sales quantities that range between 1,000 and 100,000 units.

4. In the area containing all available functions, navigate to and select the required one. You can use the filter to quickly find the function you need.

5. If you want to load functions from JavaScript Modules, click the **Add JavaScript Modules** button and add the modules with functions in the JavaScript Modules window. The loaded functions become available in the JavaScript Modules node.

    **Tip**: You can delete JavaScript modules from queries in the JavaScript Modules window.

6. **Build the expression**

    In the **Expression** window, enter the syntax for the expression to be used to calculate the contents of the column.

    The expression that you build can use data from other columns, constants and functions. You must adhere to a specific set of syntax rules when building the expression for the calculated column.

    The expression language supports the following rules:
    • constant expressions defined as variables sourced on other columns (column 1, column 2, for example) in the result set
    • numeric operators (such as +, -, *, /), character operators (+, CONCAT)
    • numeric and character functions (such as MIN, MAX, AVG, SQRT, CONCAT, SUBSTR)
    • a set of logical and relational operators (such as , , =, !=).

    The expression used to calculate the value of the added column is saved with the query.

7. Click **OK** to close the Calculated Column window.

**Results**

The content of the column is calculated according to the specified expression and added to the query result set.

If you selected **Add calculated column**... from the pop-up menu, the new column is added to the as the last column in the query results.

If you selected **Add calculated column before**... from the pop-up menu, the new column is inserted in the query results before the selected column.

If you selected **Add calculated column after**... from the pop-up menu, the new column is inserted in the query results after the selected column.

**Adding custom functions**

You can add JavaScript functions to DataQuant for Workstation and DataQuant for WebSphere and use them for building different expressions, for example, when adding a calculated column or creating a dynamart filter.

**About this task**

You can create the functions.js file or use JavaScript modules from repository to add custom JavaScript functions.
The following topics describes the detailed procedures of adding custom functions:

**Adding custom functions from a file**

You can create a file with JavaScript functions and add them to DataQuant for Workstation and DataQuant for WebSphere. Use the functions for building different expressions, for example, when you add a calculated column or create a dynamart filter.

**About this task**

To create and add a custom function:

**Procedure**

1. Create a file called *functions.js* with the description of JavaScript functions and their implementation.
2. Start each function with the keyword `'function'`.
3. Add a comment before each function.
   
   The application interprets this comment as the description of the function.
4. Optional: Define a category to which you add a new function.
   
   Use the `'@category'` tag in the comments to add it. If there is no such category, it will be created.

   **Note:** The following categories already exist in the application:
   - Columns
   - Aggregation
   - Conversion
   - Date and Time
   - Information
   - Logical
   - Math and Trigonometric
   - Misc
   - Text
   - Objects

5. Place the *functions.js* file in the DataQuant for Workstation or DataQuant for WebSphere user home directory. If you want to share *functions.js* with other users, they will have to copy the *functions.js* file to their home directories.
6. Launch or restart your application.

**Results**

The function is added to the defined category and can be used for building expressions for calculated columns, dynamart filters, and scheduler.

This allows you to implement various features in other DataQuant objects such as visual dashboards. For example, you can display data retrieved from a table on a map according to address information from that table. For more information, see “Displaying data on a map according to address information retrieved from a query” on page 301.

**Adding custom functions from repository**

You can use the functions from JavaScript modules for building different expressions in calculated columns.
Before you begin

You must have a JavaScript module with functions in the repository. For more information about JavaScript modules, see “Creating JavaScript Modules” on page 306.

About this task

To add functions from JavaScript modules:

Procedure

1. Open the query to that you want to add a JavaScript module with the functions in the query editor.
2. From the main menu, select Query > Manage JavaScript Modules. The JavaScript Modules window opens.
3. Click the Add JavaScript Module button, the Open from Repository window opens.
4. Select the JavaScript module that you want to add, click Finish. The module is added and the available functions are listed in the Available Functions list.

Results

You added a JavaScript module to a query and can use the module functions in calculated columns. The functions from the added JavaScript modules are available in the JavaScript Modules node of the Calculated Column window.

Applying categorization to query results

You can apply categorization to the calculated columns in results of visual queries. Use categorization to stratify the data and organize it into categories for effective work.

Before you begin

Open a query to which results you want to apply categorization on the Design tab of the Query editor.

About this task

To add categorization:

Procedure

1. In the Transformations pane, click the New Categorization button to open the Categorization dialog.
2. Enter the name that you want to use for the categorization in the Categorization Name field.
3. Click Add Conformance Rules Set to open the Conformance Rules dialog and specify the rules that you want to use to include the appropriate values of the result set in the category.
4. Click Add Category to open the Category dialog and specify the expressions for categorization.
5. In the Categorization dialog, select the presentation for the values that are not included in the category.
• Select **Left Out** to display empty values.
• Select **Grouped Together** and in the **Group name** field specify the name that you want to display for all values that are not included in the category.
• Select **Listed Individually** to display the values of the result set.

6. Click **OK** to close the Categorization dialog. The categorization that you created is added to the **Categorizations** node in the **Transformations** pane.

### Results

You can use the categorizations that you created to display specific data in the calculated columns of the query results.

### Adding conformance rules for categorization

You specify conformance rules to include the appropriate values of the result set in the category.

#### About this task

To add conformance rules:

**Procedure**

1. In the Categorization dialog, click **Add Conformance Rules Set** to open the Conformance Rules dialog.
2. In the **Category Name** field, enter the name that you want to use for the result set values that are included in the category.
3. Click **Add** to add a rule to the **Rules of Category** field.

   **Note:** You can add multiple rules to the same category.

4. Select the type of the rule from the **Type** area.
5. To specify the condition under which the rule is executed, select the operator from the **Operator** list and specify the required values in the **Value1** and **Value2** fields.

   **Note:** The availability of the **Value2** field depends on the operator that you selected.

6. To change the order of the specified rules, use the **Up** and **Down** buttons.
7. When you finished specifying the rules, click **OK** to return to the Categorization dialog.

### Adding expressions for categorization

You specify expressions to categorize the values of the result set.

#### About this task

To add expressions for categorization:

**Procedure**

1. In the Categorization dialog, click **Add Category** to open the Category dialog.
2. In the **Category Name** field, enter the name that you want to use for the result set values that are included in the category.
3. In the Expression for rules field, specify the expression that you want to use to categorize the values of the result set. Use the expression designer functions that are on the right of the Expression for rules field.

4. When you specified the expression, click OK to return to the Categorization dialog.

---

**Applying grouping and aggregation to query result columns**

Grouping and aggregation options can be applied to query result columns to organize the result data into logical or summarized groupings.

**About this task**

By adding grouping and aggregation you can automatically obtain summary information about your data and display the data more logically. For example you can roll up data in a report by department or average departmental commissions.

To apply grouping and aggregation to the query result columns:

**Procedure**

1. Access the menu of grouping and aggregation options in one of the following ways:
   - With query results in the available editor, select Results > Grouping and Aggregation.
   - If you are using DataQuant for Workstation, on the Results page of the editor, right-click a column's header cell in the query results and select Grouping and Aggregation from the pop-up menu.
   - From the Layout page of the editor, right-click a column name in the Layout Structure tree and select Grouping and Aggregation from the pop-up menu.

2. From the Layout page of the available editor, right-click a column label field from the Layout structure tree. A pop-up menu opens for the selected column on the right.

3. Select the grouping and aggregation option that you want to apply to the column.

4. Repeat the process for each column of data in the query results to which you want to add grouping and aggregation.

**Grouping and aggregation fields**

Grouping and aggregation options can be applied to query result columns to organize the result data into logical or summarized groupings.

**No aggregation**

Select this option to remove any aggregation or summary formatting from the selected column.

**Top Group**

Select this option to group data across the top of the query results by the selected column. Placing a column in a top group will display related values of that column on the horizontal axis, which is known as pivot or ACROSS functionality. You can define multiple columns of top groups. You can specify that Top Groups include summary information. With the column selected, click Show Total Summaries from the pop-up menu. The
query results will be reformatted so that the column spans the other columns in the result set. A summary line for each row in the query results is added.

**Side Group**
Select this option to group data down the left side of the query results by the selected column.

**Count**
Select this option to create a summary row at the bottom of the query results showing the number of values in the column.

**First**
Select this option to create a summary row at the bottom of the query results showing the first value in the column.

**Last**
Select this option to create a summary row at the bottom of the query results showing the last value in the column.

**Maximum**
Select this option to create a summary row at the bottom of the query results showing the maximum value in the column.

**Minimum**
Select this option to create a summary row at the bottom of the query results showing the minimum value in the column.

**Sum**
Select this option to create a summary row in the query results showing the sum of the values in the column. This option calculates interim summaries and overall summaries if another column is selected for grouping with summaries.

**Cumulative sum**
Select this option to display the values in the column as a cumulative sum, which is a rolling sum. Each value is added to the value in the row below it and the sum of the two values becomes the value that is added. A cumulative summary is presented at the end.

**Average**
Select this option to create a summary row showing the average of the values in the column.

**Standard Deviation**
Select this option to create a summary row showing the standard deviation of the values in the column. The standard deviation is a statistic that tells you how tightly all the various examples are clustered around the mean in a set of data.

**Percentage of group**
Select this option to display the values in the column as a percentage of the group’s total, with a summary row showing the sum of the percentages. **Percentage of group** identifies how each value in the group contributes to the whole. For example a salary of $1,000 is 10% of a total salary of $10,000.

**Percentage of total**
Select this option to display the values in the column as a percentage of the column total, with a summary row showing the sum of the percentages.

**Cumulative percentage of group**
Select this option to display the values in the column as a percentage of
the group total. A summary row at the bottom of the query results shows the sum of the percentages. For example, DEPT 42 earns 0.42% in commissions.

**Cumulative percentage of total**
Select this option to display the values in the column as a percentage value of the group total and a percentage value of the column total. A summary row at the bottom of the query results shows the sum of the percentages. For example, within DEPT 42, Wheeler earns 44.60% of the commissions, while Williams earns 55.40%, and Smith earns no commission. Overall, this DEPT 42 earns 5.75% in commissions for the company.

---

**Exporting query results**

You can export query results data to a database or to a file.

**About this task**

You use the Export Query Results wizard to export query results.

To export query results:

**Procedure**

1. Open the Export Query Results wizard in one of the following ways:
   - With a set of query results in available editor, select **Results > Export**.
   - Select **File > Export**. Select **Result set** from the list of available export wizards.
2. The Export Query Results wizard opens with the **Export query results** page.
   - If you opened the Export Query Results wizard using **Results > Export**, the name of the active query results that you will export is listed in the **Source** field.
   - If you opened the Export Query Results wizard using **File > Export**, then you must select the set of query results that you will export from the **Source** field.
3. Select where you want to export the query result data by choosing one of the following:
   - Click **File system** to export the query results data to a file. The fields that are required to export query results to a file system are displayed.
   - Click **Database** to export the query results data to a database. The fields that are required to export query results to a database are displayed.

**Sending query results in an email**

You can email query results using the Internet Mail Wizard.

**About this task**

This task describes how to send query results in an email.

**Procedure**

1. With a set of query results in active editor, select **File > Send To > Mail Recipient**...
   - The Export Query Results wizard opens with the **Select an export file type** page.
2. Select from the file types that display and click **Next**.
The second page of the Export Query Results wizard opens.

3. Select additional options for the file type and click Finish.

   Note: Available options depend on the file type you have selected. For more information, see Specifying file export options.

The Internet Mail Wizard opens.

4. On Message parameters page, specify values for the following parameters:
   - Select the format of the body text from the Format list. The available formats are Text and HTML.
   - The address to that the email must be sent in the To field.
   - The address to that a carbon copy of the email must be sent in the Cc field.
   - The subject of the email in the Subject field.
   - The body text of the email in the Message field.
   - The address from that the email must be sent in the From field.

5. To specify SMTP server parameters, click SMTP Settings. In the SMTP Settings window, specify whether to use the server that is predefined by administrator or a custom one.
   a. If you decide to use a predefined SMTP server, select the server from the list of predefined servers.
   b. If you decide to use a custom server, specify server parameters: its address and port number. If the SMTP server requires authentication, select Use SMTP authentication, and then specify user name and password. If it is necessary to specify extra access preferences, click Advanced to open the Advanced SMTP Settings window where you can add keywords that are supported by javamail. Click OK.

   Click OK to close the SMTP Settings window.

6. Click Next to add another attachment

7. Click Click here to add new file label

8. Click the ellipsis (...) and select the necessary file from your hard disk drive.

   You can attach images as well as files of other available types.

   If you want to embed an image in the body text of the email, you must add the following HTML-tag to the area of the text in which you want the image to display:
   
   `<img src="cid:[image name]">`
Where \texttt{cid:[image name]} identifies the image file you are embedding. For example, if you wanted to embed a .gif file named \texttt{chartx.png}, you would insert the following HTML-tag into the text:

\begin{verbatim}<img src="cid:chartx.png"></img>
\end{verbatim}

9. Click \textbf{Finish} to send the message.

**Exporting query results to a database**

You can export query results data to a database using the Export Query Results wizard.

**About this task**

To export query results data to a database:

**Procedure**

1. To open the Export Query Results wizard, select \texttt{File} > \texttt{Export}. Select \texttt{Result set} from the list of available export wizards and click \texttt{Next}.

2. Select \texttt{Database}. The fields that are required to export query results to a database are displayed.

3. Specify the database where you want to save the query results data in the \texttt{Database} field. When you select a database, the \texttt{Table space} field is updated to reflect the table space names that can be used.

4. Specify the owner of the table where you want to save the data in the \texttt{Table owner} field. Specify the name of the table where you want to save the data in the \texttt{Table name} field. You must use the correct format for identifiers when entering the owner or name of the table.

5. Optional: If you want to attach a comment to the data, type the comment in the \texttt{Comment} field.

6. Specify a table space name.

A default table space name that has been specified in the resource limits for your user ID is displayed when you select a database. This table space name will be used unless you specify another table space name in the \texttt{Table space} field.

You can only specify a new table space name if the table that you are adding data to does not already exist. In addition, the resource limits for your user ID must grant you permission to specify or change the table space name.

7. Click \texttt{Next}. The Set up save options page opens.

8. You can specify how much of the data in the current query results will be exported by selecting one of the following:

   - \textbf{All} to specify that all of the data in the current results grid will be saved with all the changes applied to it.
     For example, if you remove a column from the layout, it will not be saved in the export file.

   - \textbf{All - Result set only} to specify that all of the data in the original query result set not depending on the changes applied to results grid will be saved.
     For example, if you remove a column from the layout, it will be saved in the export file anyway.

\textbf{Note}: Calculated columns are exported only for visual and analytical queries.
• **Selection** to specify that only the data that you have selected in the current query results will be saved. This can include calculated columns if they are selected. The query result data that you want to save must be selected before you open the Export Query Results window.

**Note:** All - Result set only is the only available option when you export a query result set from a forecast.

9. Specify the method by which the data is saved by selecting one of the following:
   • Select **Regular - Send retrieved data back to data source and save (using an insert statement for each row)** to select the regular save mode.
   • Select **Fast - Directly save results at the data source by re-running the query at the data source** to select the fast save mode that retains any ORDER BY clause(s).

   Click this radio button to perform a fast save data process where the query is rerun at the database and the results of the query are inserted directly in the specified table.

   Not all database servers are able to perform a fast save of data. Check with your system administrator if you receive errors when saving with this method. This method retains any ORDER BY clauses.

   When the fast save method is selected, the **Remove ORDER BY clause(s) from original SQL before re-running query** check box becomes available.

   Select the check box to invoke the fast save mode that strips the ORDER BY clause(s) from the query.

10. Specify how existing data is to be handled by selecting one of the following:
   • **Replace any existing data** to replace any existing data that is in the specified table on the database.
   • **Add to any existing data** to add to any existing data that is in the specified table on the database.

11. You can choose to export any calculated columns that have been added to the query results by checking the **Save calculated columns** check box.

12. Specify a value in the **Commit scope** field that will represent the number of rows to insert before committing changes. If this value is omitted or zero, all of the rows are inserted before a commit occurs.

13. Click **Finish** to export the query result data. The Export Query Results wizard closes.

---

**Exporting query results to a file**

You can export query results data to a file system using the Export Query Results wizard.

**Before you begin**

To export a query result set to Microsoft Excel correctly, start each Excel formula with “=” in the query.

For example,

```
SELECT 45.5 AS PRODPRICE, 30 AS AMOUNT, '=A2*B2' AS PRICE
FROM SYSIBM.SYSDUMMY1
```
About this task

To save query results to a file:

Procedure

1. Open the Export Query Results wizard in one of the following ways:
   - With a query result set opened in the editor, select File > Export to open the
     Select page of the Export wizard, and then select Result set from the list of
     available export wizards.
   - With a query result set opened in the editor, click the Export (→) toolbar
     button.

2. From the Source list, select the result set that you want to export. If the query
   you want to export contains several result sets, you can export all of its result
   sets in one Microsoft Excel file by selecting the All result sets option. Each of
   the query’s result sets will be saved in a separate Excel worksheet.

   Note: Visual queries do not support several result sets.

3. If you are using DataQuant for Workstation, specify a name for the file that
   you are creating in the File name field. By default, the file is saved in your
   home directory.

4. Specify the type of file that you will be creating. Select from the list of available
   file types in the Export type field. You can choose one of the following file
   types:
   - PDF (*.pdf)
   - CSV (*.csv)
   - IXF (*.ixf)
   - XML QMF Markup Language file (*.xml)
   - HTML (*.htm)
   - TEXT (*.txt)
   - dbase III (*.dbf)
   - XLS Microsoft Excel 97-2003 (*.xls)
   - XLSX Microsoft Excel 2007 (*.xlsx)

5. If you selected any option different form XLS or XLSX, click Next. The Second
   page of the Export Query Results wizard opens. On this page you can select
   additional options that are available depending on the type of file that you are
   creating. For example, the wizard brings up a box that allows you to enter
   options for those grayed out choices when you chose DATAFORMAT=TEXT.

6. Click Finish. If you are using DataQuant for Workstation, a save window
   specific to your operating system opens. If you are using DataQuant for
   WebSphere, a save window opens that lists the available files for download.
   Click a hyperlink to download the file to your workstation.

Specifying file export options

You must specify one or more of the following options when you export query
results to a file. The options that you specify depend on the type of file that you
are exporting.
Table 14. Text options

<table>
<thead>
<tr>
<th>Field</th>
<th>File Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>What Data</td>
<td>.txt</td>
<td>You use the following radio buttons to specify what data in the query results will be saved:</td>
</tr>
<tr>
<td></td>
<td>.html</td>
<td>• All to specify that all of the data in the current results grid will be saved with all the changes applied to it.</td>
</tr>
<tr>
<td></td>
<td>.csv</td>
<td>• All - Result set only to specify that all of the data in the original query result set not depending on the changes applied to results grid will be saved.</td>
</tr>
<tr>
<td></td>
<td>.ixf</td>
<td>For example, if you remove a column from the layout, it will not be saved in the export file.</td>
</tr>
<tr>
<td></td>
<td>.wqml</td>
<td>• Selection to specify that only the data that you have selected in the current query results will be saved. This can include calculated columns if they are selected. The query result data that you want to save must be selected before you open the Export Query Results window.</td>
</tr>
<tr>
<td></td>
<td>.xml</td>
<td>Note: Calculated columns are exported only for visual and analytical queries.</td>
</tr>
<tr>
<td></td>
<td>.dbf</td>
<td>For example, if you remove a column from the layout, it will be saved in the export file anyway.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• ISO to specify that the date and time will be formatted in the file based on the ISO format. The date will be formatted as (YYYY-MM-DD) and the time will be formatted as (HH:mm:SS).</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• USA to specify that the date and time will be formatted in the file based on the USA format. The date will be formatted as (MM/DD/YYYY) and the time will be formatted as (HH:mm PM/AM).</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• JIS to specify that the date and time will be formatted in the file based on the JIS format. The date will be formatted as (YYYY-MM-DD) and the time will be formatted as (HH:mm:SS).</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• EUR to specify that the date and time will be formatted in the file based on the EUR format. The date will be formatted as (DD.MM.YYYY) and the time will be formatted as (HH:mm:SS).</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Custom to specify a custom format for date and time values. &quot;Date&quot; and &quot;Time:&quot; text fields accept Java format strings. (See <a href="http://java.sun.com/j2se/1.4.2/docs/api/java/text/SimpleDateFormat.html">http://java.sun.com/j2se/1.4.2/docs/api/java/text/SimpleDateFormat.html</a> for more details.)</td>
</tr>
<tr>
<td>Include Column Headings</td>
<td>.txt</td>
<td>Select this check box to include the query results column headings as the first row in the file.</td>
</tr>
<tr>
<td></td>
<td>.html</td>
<td></td>
</tr>
<tr>
<td></td>
<td>.csv</td>
<td></td>
</tr>
<tr>
<td>Date or Time Format</td>
<td>.txt</td>
<td>Select one of the radio buttons to specify how the date and time should be formatted in the file:</td>
</tr>
<tr>
<td></td>
<td>.html</td>
<td>• ISO to specify that the date and time will be formatted in the file based on the ISO format. The date will be formatted as (YYYY-MM-DD) and the time will be formatted as (HH:mm:SS).</td>
</tr>
<tr>
<td></td>
<td>.csv</td>
<td>• USA to specify that the date and time will be formatted in the file based on the USA format. The date will be formatted as (MM/DD/YYYY) and the time will be formatted as (HH:mm PM/AM).</td>
</tr>
<tr>
<td>CCSID</td>
<td>.txt</td>
<td>Use this field to select the code page to use when saving this file.</td>
</tr>
<tr>
<td></td>
<td>.html</td>
<td></td>
</tr>
<tr>
<td></td>
<td>.csv</td>
<td></td>
</tr>
<tr>
<td></td>
<td>.ixf</td>
<td></td>
</tr>
<tr>
<td></td>
<td>.dbf</td>
<td></td>
</tr>
</tbody>
</table>
Table 14. Text options (continued)

<table>
<thead>
<tr>
<th>Field</th>
<th>File Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Display null values</td>
<td>.txt</td>
<td>Use this field to specify the string that will be used to display null values.</td>
</tr>
<tr>
<td>Separate columns</td>
<td>.txt</td>
<td>Use this field to specify the string that will be used to separate the columns.</td>
</tr>
<tr>
<td>Delimit values</td>
<td>.txt</td>
<td>Use this field to specify the string that will be used to delimit values.</td>
</tr>
<tr>
<td>When should values be</td>
<td>.txt</td>
<td>Use this field to specify when values should be delimited. Your choices are:</td>
</tr>
<tr>
<td>delimited</td>
<td></td>
<td>• Never</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Only when necessary</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Always</td>
</tr>
<tr>
<td>Which type of values</td>
<td>.txt</td>
<td>Select one or more of the following check boxes to specify what types of values should be delimited in the file:</td>
</tr>
<tr>
<td>should be delimited</td>
<td></td>
<td>• Column headings</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Character values</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Numeric values</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Date/time values</td>
</tr>
<tr>
<td>Format</td>
<td>.ixf</td>
<td>Select one of the following radio buttons to specify how to format the .ixf output file. Your choices are:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• PC/IXF (Version 1) to export data in PC/IXF Version 1 format.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• PC/IXF (Version 2) to export data in PC/IXF Version 2 format.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• System/370 IXF to export data in System/370 IXF format.</td>
</tr>
<tr>
<td>Add Document Information</td>
<td>.pdf</td>
<td>Select this check box to add identification information in the document's properties fields when the PDF is created. Document information provides basic data about the document as well as criteria to facilitate searching. The document information includes:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Title specifies the title of the PDF document.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Author specifies the author of the PDF document.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Subject specifies the subject of the PDF document.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Keywords Specifies the keywords for the PDF document. Keywords are used as search criteria.</td>
</tr>
<tr>
<td>Use only ANSI Fonts</td>
<td>.pdf</td>
<td>Select this check box to specify that only ANSI fonts will be used in creating the PDF. If the PDF document includes only English characters, select this check box to produce a smaller PDF in less time.</td>
</tr>
</tbody>
</table>

Importing query results from CSV and text files

You use the CSV File Import wizard to import CSV and text files to the data source.
About this task

CSV and TXT files are imported into the active editor where they can be saved as result sets to the repository or QMF Catalog. To import a CSV or TXT file:

Procedure
1. Select File > Import. The Import wizard opens.
2. Select CSV File and click Next. The Import result set from file (*.csv, *.txt) page of the Import wizard opens.
3. Click the Browse next to the File name field. Select either CSV file (*.csv) or Text file (*.txt) in the Open window to choose a CSV or TXT file to import.
4. Specify what encoding to use when importing the file from the Encoding drop-down list.
5. Click Next. The Select separator options page opens.
6. Specify the separator options of the CSV or TXT file and click Next. The Result set structure page opens.
7. Specify the result set's column options in the Field definitions table and click Finish. The CSV File Import wizard closes and the new result set opens in the editor window.

Specifying separator options for CSV and TXT files
You use the Select separator options page of the CSV File Import wizard to specify how DataQuant will convert the data of the CSV or TXT file into the columns of the result set.

About this task

The separator options of the CSV File Import wizard allows you to specify how DataQuant interprets the format of the CSV or TXT file to convert it into a new result set, and how the finished result set will look. To specify separator options for the CSV or TXT file:

Procedure
1. Open the Select separator options page of the CSV File Import wizard.
2. Select the Skip rows starting with check box and specify a string in the text field to exclude certain rows from the result set.
3. Select what symbol is used to display decimal points in the file in the Decimal separator drop-down list.
4. Select the First row contains column names check box to specify that the first line of the file contains the names of columns to be created.
5. To specify that the columns will be created based on a fixed set of character lengths, click Fixed width and then specify the specific widths of the columns by entering the character lengths in the Specify columns width field. When each character length is reached in the text, the characters after that point are placed in a separate column, regardless of the content of the text.
6. To specify that the columns will be created automatically by breaking at certain symbols, click Separated by and then selecting the symbols check boxes below. When the result set is made, each time these symbols are encountered in the text, the characters after the symbol are placed in a separate column.
7. Select the Merge separators check box to consolidate columns that are created by multiple adjacent separators into one column. For example, if Comma and Space are selected and the text includes the string NAME, DEPT, three columns
will be created. The first column contains NAME, and was created when the
comma was encountered. The second column is empty, and was created when
the space was encountered. The third column contains DEPT, and was created
when the remaining text was encountered. By selecting the Merge separators
check box, only one column is created when the comma and the space are
encountered.

8. Select whether a single quote or a double quotation marks are used to delineate
text in the Text qualifier drop-down list. Text that is contained in the specified
quotes is always displayed as is, whether it contains a separator symbol or not.

Note: You can track how each selection changes the layout of the result set in
the Preview table.

Specifying the result set structure of CSV and Text files

You use the Result set structure page of the CSV File Import wizard to customize
the columns of the result set, and which columns are displayed.

About this task

By specifying the options in the Field definitions table, you can customize the
structure of the imported result set:

Procedure

1. Select what columns to include in the result set by checking the check boxes to
   the left of the Field name column.
2. Edit the name of a column by clicking a cell in the Field name column and
   entering the new name.
3. Specify the type of a column by clicking a cell in the Type column and
   specifying a new column type from the drop down menu. Available options are
   the following:
   - INTEGER can be used to store signed whole numbers between -2,147,483,648
     and 2,147,483,647.
   - BIGINT can be used to store large signed whole numbers between -9 223
     372 036 854 775 808 and 9 223 372 036 854 775 807.
   - DOUBLE can be used to store large numbers with a floating decimal point.
   - CHAR is used to store fixed-length string of characters. For example, you
     can use it to store employee phone number which is generally fixed in
     length.
   - VARCHAR is used to store variable-length string of characters like employee
     name.
   - DECIMAL is used to store the fixed precision and scale numbers. This data
type should be used for precise values, such as currency.
   - DATE is used to store a date only.
   - TIME is used to store a time only.
   - TIMESTAMP is used to store both a date and a time.

Note: DATE, TIME, and TIMESTAMP data types support different format
options that define how the values are displayed in the result set.

Note: The supported ranges and the precision of values can vary depending on
the database you use.
4. Specify the length of a column’s character set by clicking a cell in the **Length** column and specifying a new length from the list.

**Note:** You can track how each selection changes the layout of the result set in the **Preview** table.

5. Click **Finish**. The CSV File Import wizard closes and the new result set opens in the editor window.

---

**Importing data contained in IXF files**

Data contained in IXF files can be directly opened in the application.

**Opening the contents of an IXF file for further processing**

Once opened, the contents of the IXF file are presented in a results view, much the same as the output from a query.

**About this task**

The results view then allows you to format the data, generate a report, export it to a file or database table, offering options identical to those available when viewing query results.

**Procedure**

1. Click the Open from file toolbar button and enter the IXF file name. The data contained in the IXF file opens as a results view in the editor.

2. Use the **Results** menu to run the following commands:
   - **Font** opens the Font page of the Layout Properties window where you can view and change the font attributes of your selection within the query results.
   - **Format** opens the Format page of the Layout Properties window where you can view and change the formatting options for the selected column in the query results.
   - **Grouping and Aggregation** specifies logical groupings of information within the query results.
   - **Show Total Summaries** shows summary information for the selected total columns.
   - **Add Calculated Column** opens the Calculated Columns window enabling you to add a calculated column to the query results. A calculated column is a column of data that is dynamically created based on an expression that you define.
   - **Reset Formatting** resets the formatting that has been applied to selected columns in the query results to the original formatting attributes.
   - **Reset All** resets the formatting that has been applied to all the columns in the query results to the original formatting attributes.
   - **Autofit** automatically fits the contents of columns and rows.
   - **Sort** opens the Sort window where you can set up multiple levels of sort criteria.
   - **Display Report** generates a visual or classic report using the query results that are active in the editor.
   - **Display Excel Sheet** opens Microsoft Excel and displays the query results data in an Excel data sheet.
• **Export** opens the Export Query Results wizard where you can choose to save the query results to a file or a database.

**Directly importing an IXF file into a database table**

You can also directly import the contents of an IXF file to a database table.

**About this task**

To import data contained in IXF files into a database table:

**Procedure**

1. Select **File > Import**. The Import wizard opens.
2. Select **Table** and click **Next**. The Import table from file (*.ixf, *.xml, *.csv, *.txt, *.dbf) page of the Import Table wizard opens.
3. Click **Browse** next to the **File name** field and enter the IXF file name.
4. Click **Next**. The Select results and destination information page of the Import Table wizard opens.
5. Specify the table owner name in the **Table owner** field.
6. Click **Next**. The Set up save options page of the Import Table wizard opens.
7. Specify **All** for the **What data?** option. Since you are retrieving data from a file rather than a query, all rows and columns will be imported.
8. Specify **Regular** for the data source **Method** option. Regular uses an insert statement for each row.
9. Specify the **Existing Data** option. You use the following radio buttons to specify how the existing data will be saved to the database table:
   - **Replace any existing data** to replace any existing data in the database table with the contents of the IXF file.
   - **Add to any existing data** to append the IXF file contents to any data that already resides in the database table.
10. Click **Finish**. The Import Table wizard closes and the data contained in the IXF file is saved to the database table.

**Importing table contents of a saved file into a database table**

You can import the table contents that are saved in a file into a database table.

**About this task**

To import data contained in a file into a database table:

**Procedure**

1. Select **File > Import**.
2. From the list of available wizards select **Table** and click **Next**.
   
The Import table from file page of the Import Table wizard opens.
3. Click **Browse** next to the **File name** field and specify the file name and location.
   
The following file formats are supported:
   - *.txt
   - *.csv
   - *.ixf
• *.xml
• *.dbf
4. Click Next.
   If you import the table contents from *.txt or *.csv files, the CSV File Import page opens.
   a. From the Encoding drop-down list, specify the type of encoding to use when importing the file.
   b. Click Next
      The Select separator options page opens.
   c. Specify the separator options of the CSV or TXT file.
5. Click Next.
   The Select results and destination information page opens. Specify the table owner name in the Table owner field.
6. Click Next
   The Set up save options page of the Import Table wizard opens.
7. Specify All for the What data? option.
   Note: Since you are retrieving data from a file rather than a query, all rows and columns are imported.
8. Specify Regular for the data source Method option.
    Regular uses an insert statement for each row.
9. Specify the Existing Data option.
    Use the radio buttons to specify how to save the existing data to the database table:
    • Replace any existing data to replace any existing data in the database table with the contents of the import file.
    • Add to any existing data to append the import file contents to any data that already resides in the database table.
10. Click Finish.

Results

The Import Table wizard closes and the data contained in the file is saved to the database table.

Saving data objects as dynamarts

Dynamart is a format for saving result sets with all the modifications.

About this task

When you save a data file as a dynamart, the link to this file is saved within it. When you open the dynamart, it reloads the data from the linked source file and updates the result set if the source file was modified or replaced. You can change the source object or deactivate the data update for a particular dynamart in the Dynamart Settings window.

When you save a query as a dynamart, the result set and the operations with it are saved within the dynamart. When you open the dynamart, it reloads the result set and automatically applies all the custom changes, like filtering or adding calculated columns.
To save data objects as dynamarts:

**Procedure**

1. Make sure that the data file that you want to save as a dynamart is opened in the active window. If you work with an analytical or visual query, run it before you save it as a dynamart.

   **Note:** To convert a query to a visual query, click the Convert Query to Visual Query toolbar button

2. From the main menu, select **File > Save At**.

3. Select one of the following options depending on where you want to save the dynamart:
   - **Save to Repository**
   - **Save to File**

4. In the Save to Repository window, specify the parent entry, file name, and comment.

5. In the Save to File window, specify the directory to save the file and the file name. You can click **Browse** to search the destination directory.

6. Select one of the following options from the **Type** list:
   - **Dynamart** to save a visual query, CSV, DBF, IXF, TXT, or XML files.
   - **Compound Dynamart** to save an analytical query.

7. Click **Finish** to save the dynamart files.

**Managing source data files in dynamarts**

After saving the dynamart, you can change or completely remove the link to the source data object.

**About this task**

When you save data files as dynamarts, the link to this data file and the time of its last modification are saved within the dynamart. When you open this dynamart, DataQuant reloads the data from the linked source file and updates the result set if the source file was modified or replaced.

You can change the source object or disable data update for a particular dynamart in the Dynamart Settings window.

To replace the source data file or disable data update:

**Procedure**

1. Select **Results > Dynamart Settings**.

2. In the Dynamart Settings window, specify the name and location of a data file that you want to use as source file in the **Data file** field. You can click **Browse** to search the file.

3. Select the **Update dynamart when file is changed** check box if you want the result set to be updated on any changes in the linked data file.

4. Clear the **Update dynamart when file is changed** check box to unlink the source data file. In this case the dynamart always displays original data even if the data file is replaced or updated.

5. Click **OK** to close the Dynamart Settings window.

**Related tasks:**
Working with LOB data in the query results

When retrieving large object data you must identify the application that will be used to view the LOB data that has been included in the query results column.

About this task

Retrieving and saving large amounts of data can be time consuming. In order to prevent excessive resource consumption, certain restrictions are placed on a user's ability to retrieve and save large object data. Your ability to retrieve and save LOB data is dictated by the LOB Option resource limit set for your user ID or group by your DataQuant administrator.

To retrieve or save LOB data:

Procedure

1. Run a query or procedure that calls the table where the LOB data is stored on the database.
2. The LOB data is retrieved from the data source. The contents of each cell in the column that contains LOB data can appear differently depending on the LOB resource limit that has been set for your user ID:
   • If the LOB Retrieval Option resource limit is Retrieve LOB data automatically, then all the LOB data is available to you and displayed in the query results.
     This option pulls ALL of the LOB data for ALL of the LOB columns from the database to the local computer. The actual LOB data is not displayed in grids and reports. Instead, pointers to the LOB data are displayed.
     Note: This option can potentially consume a large amount of resources on the local computer.
   • If the LOB Retrieval Option resource limit is Retrieve LOB data on demand, then each cell in the column containing LOB data is identified with the label, LOB locator. You must initiate the retrieval of the of the LOB data.
3. To initiate the retrieval of the LOB data, double-click the cell labeled LOB locator. The LOB Type Associations window opens. You can also open the LOB Type Associations window by right-clicking in the header cell of the query results column that contains the LOB data and selecting LOB Type Associations from the pop-up menu.
4. From the LOB Type Associations window you will identify the type of LOB data that is included in each of the cells of the query results column. The application that will be used to open the LOB data file will be determined based on the file extension that you select. You can select one of the following:
   • Select File Extension to specify the type of LOB data file that will be included in each cell of the query results column. Choose from the drop-down list of available extensions. This option will apply the extension that you select to all cells in the query result column.
   • Select Mapping Column to individually specify which application will be mapped to each LOB data cell in the query results column. From the
drop-down list, select the column in the query results that contains the file extension information for the LOB data. Click Mapping. The File Extension Mapping window opens.

a. The Mapping table includes the following fields:
   - Mapping Value: Lists each unique data type that exists in the mapping column.
   - File Extension: Contains the file extension of the unique data type, and the application that will be used to open the LOB data.

b. Click the File Extension field. Type the file extension you want to represent the unique data type. For example, if the mapping column in the query results contains the text string "bitmap", you would type a file extension of .bmp. You can also select from a drop-down list of pre-defined file extensions and their associated applications that will be used to open that type of file.

c. When all file extensions have been specified, click OK. Control returns to the LOB Type Associations window.

5. Click OK. The Column LOB Type Associations window closes.

6. To retrieve and view the LOB data you double-click LOB data's cell. The LOB data opens in the designated application. In the query results grid, the label for the cell will change from LOB locator to LOB value to indicate that the LOB data has been retrieved. You can Select Retrieve all LOB values to retrieve all large object data files that are included in the query results column.

**Overriding LOB resource limit option**

If the DataQuant administrator has granted your user ID permission, you can override the LOB resource limits that have been set for your resource limits group.

**About this task**

To override the LOB retrieval resource limit:

**Procedure**

1. Select View > Preferences. The Preferences window.
2. Click LOBs to change the default options for retrieving LOBs.
3. Check the Override LOB options if possible check box to make changes to the LOB options. If this check box is not available, your user ID does not have permission to override LOB resource limits.

**Viewing and editing the contents of a cell in the query results**

If you are using DataQuant for Workstation, you can use the Zoom option to view in hexadecimal or binary the contents of a cell in the query results.

**About this task**

This topic describes how to:

- View the contents of a query results cell when the results are read-only. See step 1
- View and edit the contents of a query results cell when the results are writable (not read-only). See step 2
Procedure

1. To view the contents of a query results cell when the results are read-only:
   a. Right-click a cell in the query results. Select **Zoom** from the pop-up menu. The Zoom window opens.
   b. The **CCSID** field specifies the code page that was used to save the data in the selected query results cell.
   c. The current value of the selected query results cell is displayed in the **Cell value** field.
   d. Click **Advanced** to expand the dialog with additional fields to display the data in hexadecimal or binary format. This button is disabled for cells with numeric data.
   e. Select **Hexadecimal** to display the data in hexadecimal format. Select **Binary** to display the data in binary format. The Zoom window displays samples of the data in the format that you have chosen.
   f. Select **Cancel** to close the Edit window.

2. To view or edit the contents of a query results cell when the results are writable (not read-only):
   a. Right-click a cell in the query results and select **Edit** from the pop-up menu. The Zoom Edit window opens.
   b. The CCSID field specifies the code page used to save the data in the selected query results cell.
   c. The current value of the selected query results cell is displayed in the **Cell value** field. The **Cell value** field is editable.
   d. Make the necessary changes and select **OK** to apply them.
   e. Click **Advanced** to expand the dialog with additional fields to display the data in hexadecimal or binary format.
      This button is disabled for cells with numeric data.
   f. Select **Hexadecimal** to display the data in hexadecimal format or select **Binary** to display the data in binary format. The Zoom Edit window displays samples of the data in the format that you have chosen.
   g. Select **Cancel** to close the Zoom Edit window.

To edit a cell in the tables on which the query is based you can use the table editor.

Viewing multiple result sets

There are two instances where multiple query result sets might be returned. The first instance is as a result of running a stored procedure. The second instance is as a result of running a single query that has multiple SQL statements.

About this task

All result sets are returned to the editor window and assigned an index number starting with the number one (1). Each set of query results is identified in a drop-down list on the query editor toolbar. To view a specific result set, select it from the drop-down list.
Results menu

The Results menu becomes available when the results of a query are returned and are the active contents in the editor.

The following menu commands are available:

**Retrieve All**

Select Retrieve All to finish retrieving all the query results data. When retrieving large amounts of data, unless otherwise specified, only the first 100 data rows are retrieved.

**Font**

Opens the Font page of the Layout Properties window where you can view and change the font attributes of your selection within the query results.

**Format**

Opens the Format page of the Layout Properties window where you can view and change the formatting options for the selected column in the query results.

**Grouping and Aggregation**

Use the Grouping and Aggregation menu choices to specify logical groupings of information within the query results. The following options are available:

- Select **No Aggregation** to remove any aggregation or summary formation from the selected column.
- Select **Top Group** to group the selected column without summary information in the top of the query results.
- Select **Side Group** to group the selected column without summary information to the left side of the query results.
- Select **Count** to summarize the total number of values in the column.
- Select **First** to create a summary row showing the first value in the column.
- Select **Last** to create a summary row showing the last value in the column.
- Select **Maximum** to create a summary row showing the maximum value in the column.
- Select **Minimum** to create a summary row showing the minimum value in the column.
- Select **Sum** to create a summary row showing the sum of the values in the column.
- Select **Cumulative Sum** to display the values in the column as a cumulative sum, with a summary.
- Select **Average** to create a summary row showing the average of the values in the column.
- Select **Standard Deviation** to create a summary row showing the standard deviation of the values in the column.
- Select **Percentage of Group** to display the values in the column as a percentage value of the group total, with a summary row showing the sum of the percentages.
- Select **Percentage of Total** to display the values in the column as a percentage value of the column total, with a summary row showing the sum of the percentages.
• Select **Cumulative Percentage of Group** to display the values in the column as a cumulative percentage value of the group total, with a summary row showing the sum of the percentages.

• Select **Cumulative Percentage of Total** to display the values in the column as a cumulative percentage value of the column total, with a summary row showing the sum of the percentages.

**Show Summaries**
Show summary information for the selected Top or Side group column.

**Show Total Summaries**
Show summary information for the selected total columns.

**Drill Down**
Expand the Top or Side group detail information.

**Drill Up**
Collapse the Top or Side group detail information.

**Add Calculated Column**
Opens the Calculated Columns window enabling you to add a calculated column to the query results. A calculated column is a column of data that is dynamically created based on an expression that you define.

**Reset Formatting**
Resets the formatting that has been applied to selected columns in the query results to the original formatting attributes.

**Reset All**
Resets the formatting that has been applied to all the columns in the query results to the original formatting attributes.

**AutoFit**
Select **AutoFit** to automatically fit the contents of one of the following:

• **Selected Column(s)** to automatically fit the contents of the selected columns into the same column size.

• **All Columns** to automatically fit the contents of all the columns in the query results into the same column size.

• **All Rows** to automatically fit the contents of all the rows in the query results into the same row size.

• **All** to automatically fit the contents of the entire query results.

**Sort**
Opens the Sort window where you can set up multiple levels of sort criteria.

**Go to**
Opens the Go to Row window, where you can navigate to a specific row.

**Find**
Initiates a search of the query results for a specific string. You specify the search string using the Find window.

**Display Report**
Generates visual or classic report using the query results that are active in the editor. You can choose to generate the report based on the existing formatting of the query results or simpler default formatting.

**Display Excel Sheet**
Opens Microsoft Excel and displays the query results data in an Excel data sheet. (DataQuant for Workstation only). If you are exporting a query with several result sets, each of the result sets will be displayed in a separate workbook.

Chapter 8. Managing query results 133
Note: Visual queries do not support several result sets.

Note: For relational query results whose columns have both breaks and usage codes, an Excel PivotTable will be created automatically. Processing time for this functionality increases depending on the amount of data.

**Retrieve All LOB Values**
Retrieves all LOB values in the query results set.

**Export**
Opens the Export Query Results wizard where you can choose to save the query results to a file or a database.

**Display Chart**
Generates a chart from a query result set that is open in the editor.

---

### Sorting query results

Use the Sort dialog to sort the query results by up to three columns of data.

**About this task**

To specify the columns by which the query results will be sorted:

**Procedure**

1. With query results in the available window, open the Sort window by selecting **Results > Sort**.
2. Specify the first column that will be used to sort the query results using the **First Sort Rule** fields. You must select the column from the **Column** list, specify the type of sort in the **Type of sort** list, and select **Ascending** or **Descending** to specify the sort order.
3. Optionally, you can specify a second and a third column that will be used to sort the query results. You use the **Second Sort Rule** and **Third Sort Rule** fields to specify the column, type of sort and the sort order for each of the additional columns.
4. By default, the **Sort string based on locale rules** check box is clear indicating that the query results are sorted based on the comparison of characters according to their Unicode code point values. Select the **Sort string based on locale rules** check box to sort query results based on a character's position in the national language alphabet that has been defined for the current locale.
5. Click **OK**. The query results are sorted and the Sort window closes.

**Results**

The specified sorting order will be saved within the query and used the next time you run it if the DSQQW_QUERY_PRESERVE_SORT global variable is set preserve sorting order (1). If you need to rerun the query without applying the specified sorting order, set the DSQQW_QUERY_PRESERVE_SORT global variable to zero (do not preserve sort order), and then run the query. You will always be able to apply the specified sorting order from the Sort dialog.

---

### Specifying print options for query result sets and quick reports

In DataQuant for Workstation, you can specify the page setup options that will be used when you print the query, visual query or quick report opened in the editor window.
About this task

To control the appearance of printed pages for query result sets or quick reports:

Procedure

1. With an active object in the editor window, open the Page Setup window by selecting **File > Page Setup**.
2. Specify the printing options that will apply to your object.
   - The sets of options vary depending on the type of object you want to print. The following table shows what page options you can specify for query result sets and quick reports:

<table>
<thead>
<tr>
<th><strong>Query result set</strong></th>
<th><strong>Quick report</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Printer and page parameters including the type and size, orientation and margins</td>
<td>Paper parameters including the type and size, orientation and margins</td>
</tr>
<tr>
<td>Header and Footer text</td>
<td>Footer text</td>
</tr>
<tr>
<td>Whether to print row and column numbers, column headers, grid lines, aggregation type, and page header and footer</td>
<td>Whether to print row and column numbers, column headers and page footer</td>
</tr>
<tr>
<td>Page order</td>
<td>Fixed groups</td>
</tr>
<tr>
<td>Fixed side and top groups</td>
<td></td>
</tr>
</tbody>
</table>

   **Note:** You can use special codes to insert specific data into header and footer fields.

3. To make the specified values default, click **Set as Default**.
4. Click **OK** to close the Page Setup window. The options that you specified are used to control the appearance of the printed object.

Codes to insert specific data in a page header and footer

You can use special codes to insert specific data in a header or footer.

When you print a query, visual query result set or a quick report, you might want to add some data like the document name or the total number of printed pages to the page header or footer. To do this, you can use special codes that are described in the following table:

<table>
<thead>
<tr>
<th><strong>Code</strong></th>
<th><strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>&amp;f</td>
<td>Prints the name of the object.</td>
</tr>
<tr>
<td>&amp;n</td>
<td>Prints the current page number.</td>
</tr>
<tr>
<td>&amp;N</td>
<td>Prints the total number of pages.</td>
</tr>
<tr>
<td>&amp;d</td>
<td>Prints the current date.</td>
</tr>
<tr>
<td>&amp;t</td>
<td>Prints the current time.</td>
</tr>
<tr>
<td>&amp;&lt;</td>
<td>Prints the number of the first printed page.</td>
</tr>
<tr>
<td>&amp;&gt;</td>
<td>Prints the number of the last printed page.</td>
</tr>
<tr>
<td>&amp;&amp;</td>
<td>Prints a single ampersand.</td>
</tr>
<tr>
<td>&amp;</td>
<td></td>
</tr>
</tbody>
</table>
Examples

| & | &n | The current page number is printed in the center of a header (footer). |
|-----------------------------------|
| & | & | &d | The current date is printed on the right side of the header (footer). |
| f & | & | &d | The object name is printed on the left side of the header (footer); and the current date is printed on the right side of the header (footer). |

Saving data using the regular and fast save methods

When you are saving query results data to a database, you can choose to save the data using a "regular save" or a "fast save" method.

When saving your query results data using a "regular save" method, the interface that you are using (DataQuant for Workstation or DataQuant for WebSphere) saves the retrieved query results to the database using an individual SQL INSERT statement for each row of data. This type of save can be expensive in terms of performance due to the overhead of passing large amounts of data back to the database.

When saving your query results data using a "fast save" method, all the processing occurs at the database. There is no further passing of data between the DataQuant interface and the database. The DataQuant interface adds SQL to the original query, the query is rerun and the data is saved directly into the specified table(s). Saving query results data using the "fast save" process significantly improves performance when a large amount of data is to be saved.

Selecting the fast save method

If you elect to save your query results data using a "fast save" method, all ORDER BY clauses remain in the query that is run at the database. Keeping the ORDER BY clause retains the row order of the query results data.

In some instances on DB2 for z/OS systems, keeping the ORDER BY clause can cause an SQL error. This error results from the DataQuant interface prepending an insert statement to the query. The original query is now a subordinate query and thus is not permitted to have an ORDER BY clause.

For example if a user enters the following query:

Select [Columns] from [SourceTable]
Order By [Column]

The query runs without error. The user then selects to save it into a given table (e.g. DestTable) using the fast save mode. The DataQuant interface prepends a line to the query as follows:

Insert into [DestTable]
Select [Columns] from [SourceTable]
Order By [Column]
This new SQL is invalid in certain DB2 for z/OS systems because the original query is now a subordinate query and thus is not permitted to have an ORDER BY clause.

**Setting the regular or fast save method**

To set the regular or fast save method:

1. With query results in the editor window, select Export from the Results menu. The Export Query Results window opens. Select Database from the Export Query Results window. Enter values in the Export destination fields. Click Next. The Set up save options page opens.

2. Select one of the following:
   - Click Regular (send retrieved data back to data source and save using an insert statement for each row) to select the regular save method.
   - Click Fast (directly save results at the data source by re-running the query at the data source) to select the fast save method that retains any ORDER BY clause(s).
   - Select the Remove ORDER BY clause(s) from original SQL before re-running query check box to select the fast safe method that runs the query without the ORDER BY clauses. Use this option when the removal of ORDER BY clause(s) does not influence the returned data.

**Selecting the fast save method with the remove ORDER BY clause option**

If you elect to save your query results data using the "fast save" method, you can choose to have the DataQuant interface remove all ORDER BY clauses from the query before the query is sent to the database. In keeping with the example used in "Selecting the fast save method" on page 136, the following query would be sent:

```
Insert into [DestTable]
Select [Columns] from [SourceTable]
```

In most cases, removing the ORDER BY clauses has no effect on the query result data that is saved. In some cases, however, removing the ORDER BY clause from the query can affect the result set that is saved. This will happen if the original query limits the number of rows returned. For example, removing the ORDER BY clause from the following query changes the results:

```
select x,y,z from table
order by x
fetch first 10 rows only
```

**Setting the regular or fast save method for procedures**

You can set up a regular or fast save method for your procedure query results data using the global variable DSQQW_FST_SV_DA. The default value for DSQQW_FST_SV_DA is the value zero (0).


2. From the Global variables window select the DSQQW_FST_SV_DA variable.

3. Select or specify:
   - 0 - Regular save
   - 1 - Fast save with ORDER BY clause(s) stripped
• 2 - Fast save with ORDER BY clause(s)

You can also set the global variable in either of the following ways:

• Use the Set Global procedure command using either DataQuant interface. For example SET GLOBAL ( DSQQW_FST_SV_DATA = 2 ).
• You can also set DSQQW_FST_SV_DATA to 2 using SetGlobalVariables() in the API for either DataQuant interface.
Chapter 9. Working with scheduled tasks

Scheduled tasks allow you to create automated actions such as running queries and exporting query results without taking direct action from the user interface.

About this task

Scheduled tasks are invaluable tools that help to automate business processes to run at specified dates and times. This is especially useful for actions that must be run on a regular basis, such as weekly sales reports that are supposed to be distributed to several different locations, or quarterly earnings reports.

To create a scheduled task:

Procedure

1. Select View > Scheduled Tasks or File > New > Scheduled Task.
2. Click Add Task to create a scheduled task if you use Task List dialog.
3. If you use Create New Scheduled Task wizard by selecting File > New > Scheduled Task. Select the type of scheduler you want to use. You can use a local scheduler or DataQuant for WebSphere scheduler. And click Next
4. Specify a name for the task in the Task name field.
6. Specify a name for the action in the Action name field.
7. Select what type of action you want to add and click Next.
8. Specify the options for the selected task type and click Finish. The Action Wizard closes and the new action is added to the Actions list.
9. To generate a detailed log of this task execution, select the Enable logging check box. The error log and the task execution report are saved to the user home directory if the task is run from the operating system scheduler.
10. When you have added all of the actions that you want the task to run, click OK. The Task window closes and control returns to the Task List window.
11. Click Import task to import a task.
12. Click Export task to export a task.
13. To schedule a task, select the task that you want to schedule and click Schedule task.
14. Click OK. The Task List window closes.

Adding an execute object action

The execute object action allows you to automatically execute queries, a visual report, procedure, or quick report, and export, mail, or print it.

About this task

To add an execute object action to a scheduled task:

Procedure

1. Open a task in the Task List window.
2. Click Add Action. The Action Wizard opens.
3. Specify a name for the action in the **Action name** field, click **Execute object**, and then click **Next**. The Specify object page opens.

4. Select the type of object that you want to execute.

5. Specify the location of the object. There are two options:
   a. To specify the static source location of the object, click **Path** and then click the ellipsis (...) at the bottom right of the **Source** area. The Open window opens.
   b. Navigate to and select the object and click **Finish**. The Open window closes.
   c. To specify a conditional source location for the object, click **Formula** and then click the ellipsis (...). The Formula editor opens.
   d. Specify the formula to use to determine the location of the object and click **OK**. The Formula editor closes.

6. Select **Set a new data source** check box to specify a particular data source against which you want to execute the object. Then click the ellipsis (...), the Add Data Source window opens, select the required data source and click **OK**.

7. To add an available data source, click **Add Data Source**. The Add Data Source window opens.

8. Navigate to and select the data source that you want to add, specify a login name and password for the data source, and click **OK**. The Add Data Source window closes.

9. Click **Next**. The Specify operations page opens.

10. Click **Add Operation**. The Operation Wizard opens.

11. Select an operation and click **Next**.

12. Specify the options for the selected operation and click **Finish**. The Operation Wizard closes.

13. When you have finished adding all of the operations that you want the action to run, click **Finish**. The Action Wizard closes and control returns to the Task window.

14. Click **OK**. The Task windows closes and control returns to the Task List window.

15. Click **OK**. The Task List window closes.

---

**Specifying export query operation options**

You use the export query operation to export query results to a file.

**About this task**

To add an export query operation to an action:

**Procedure**

1. Open the Specify operations page of the Action Wizard and click **Add Operation**. The Operation Wizard opens.

2. Click **Export query** and then click **Next**. The Specify export parameters page opens.

3. If you export a query click the necessary result set number in the **Result set** field. If you export a visual query select the necessary display mode from the **Display mode** list if several of them are available.

4. Select the file type that you want to export from the **Export type** drop-down list. Valid file types are:
5. Specify the location to save the object. There are two options:
   a. To specify a static save location for the object, click Path and then click the ellipsis (…) at the bottom right of the Destination area. The Save As window opens.
   b. Navigate to the directory location where you want to save the object and click Finish. The Save As window closes.
   c. In case the Root output directory is specified on the Server-Side File System page of the Preferences window, you can just type a path and a file name or only a file name in the Path field. The file will be exported to this root directory automatically. In DataQuant for WebSphere you can specify only the path, lying in the root directory set by administrator. Otherwise, the export will be forbidden and an error will occur.
   d. To specify a conditional save location for the object, click Formula and then click the ellipsis (…). The Formula editor opens.
   e. Specify the formula to use to determine the location to save the object and click OK. The Formula editor closes.

6. Select the Delete after task completion check box if you want to remove the exported file from the specified location when the scheduled task is completed.

7. Select the Mark as attachment check box if you want to be able to attach this query result set to the email automatically.

8. Click Next. The Specify [file type] export options page opens. On this page you can select additional options that are available depending on the type of file that you are creating.

9. Click Finish. The Operation Wizard closes.

Related tasks:
“Setting the directory location preference for objects generated by Export actions” on page 18

You can set preferences in DataQuant so that objects generated from various DataQuant actions are exported to a specific location on either the local file system or the server file system.

Related reference:
“Specifying file export options” on page 120

You must specify one or more of the following options when you export query results to a file. The options that you specify depend on the type of file that you are exporting.

**Specifying export query to mail operation options**

You use the export query to mail operation to send query results in an email.

**About this task**

To add an export query to mail operation to an action:
**Procedure**

1. Open the Specify operations page of the Action Wizard and click **Add Operation**. The Operation Wizard opens.
2. Click **Export query to mail** and then click **Next**. The Specify export parameters page opens.
3. If you export a query click the necessary result set number in the **Result set** field. If you export a visual query select the necessary display mode from the **Display mode** list if several of them are available.
4. Select the file type that you want to export from the **Export type** drop-down list. Valid file types are:
   - Text (*.txt)
   - CSV (*.csv)
   - PDF (*.pdf)
   - HTML (*.htm)
   - IXF (*.ixf)
   - XML Extensible Markup Language file (*.xml)
   - XLS Microsoft Excel 97-2003 (*.xls)
   - PNG (*.png)
5. Specify the name of the file attachment in the email in the **Attachment name** field.
6. If you want to specify a conditional file attachment name, click the button next to the field. The Formula editor opens.
7. Specify the formula to use to determine the name of the attachment and click **OK**. The Formula editor closes.
8. Click **Next**.
   The Specify [file type] export options page opens. On this page you can select additional options that are available depending on the type of file that you are creating.
   Click **Next**.

   **Note:** The values that you set for can be specified dynamically during Iterator work with the help of substitution variables. You can use an action group with Iterator to send separate email messages to several recipients. To do so you need to specify Iterator query that returns all necessary email information, define Iterator variables, and type the variable names in the corresponding fields on the Specify mail settings page using the equality sign (=Form, =SMTP).

   When you run the task, Iterator goes over the Iterator query records, extracts a set of email parameters at each iteration, and passes it to the **Export query to mail** operation.

   - From the **Format** list, select the format of the body text. The available formats are Text and HTML.
   - Specify the recipient’s email address in the **To** field.
   - Specify additional recipients in the **Cc** field.
   - Specify the subject of the email in the **Subject** field.
   - Specify the content of the email in the **Message** field.

   **Note:**
If you want to include an embedded image in the text of the email, you must insert the following HTML snippet into the text:

```html
<img src="cid:[image name]"/>
```

where `cid:` is the standard abbreviated content identifier required when referencing an embedded image within a formatted email, and where `[image name]` is the name of the image file. For example, to embed an image file named `image1.png`, you would insert the following HTML snippet into the text of the email:

```html
<img src="cid:image1.png"/>
```

- Specify the email address from which the email is sent in the **From** field.

10. To specify SMTP server parameters, click **SMTP settings**. In the SMTP Settings window, specify whether to use a server predefined by administrator or a custom one.
   a. If you decide to use a predefined SMTP server, select the required server from the list of predefined servers.
   b. If you decide to use a custom server, specify server parameters including its address and port number. If the SMTP server requires authentication, select **Use SMTP authentication**, and then specify user name and password. If it is necessary to specify additional access preferences, click **Advanced** to open the Advanced SMTP Settings window where you can add required keywords supported by javamail. Click **OK**.

Click **OK** to close the SMTP Settings window.

11. Click **Finish** to close the Operation wizard.

**Related reference:**

“Specifying file export options” on page 120

You must specify one or more of the following options when you export query results to a file. The options that you specify depend on the type of file that you are exporting.

### Specifying print query operation options

You use the print query operation to print the specified query results.

**About this task**

To add a print query operation to an action:

**Procedure**

1. Open the Specify operations page of the Action Wizard and click **Add Operation**. The Operation Wizard opens.
2. Click **Print query** and then click **Next**. The Specify printer page opens.
3. Select the printer that you want to use to print the query results from the **Printer** drop-down list.
4. Click **Properties**. The Page Setup window opens.
5. Select the size of the printable area from the **Size** drop-down list.
6. Select the paper tray to use from the **Source** drop-down list.
7. Select the orientation of the printout from the **Orientation** radio group.
8. Specify the size of the margins of the page in the **Margins** fields.
9. Click **OK**. The Page Setup window closes and control returns to the Operation Wizard.
10. Click Finish. The Operation Wizard closes.

Specifying export quick report to mail operation options
Use the export quick report to mail operation to send a quick report in an email.

About this task
To specify export quick report to mail operation options, perform the following procedure:

Procedure
1. Open the Specify operations page of the Action Wizard and click Add Operation. The Operation Wizard opens.
2. Click Export quick report to mail and then click Next. The Specify export parameters page opens.
3. Select the file type that you want to export from the Export type drop-down list. Valid file types are:
   - PDF (*.pdf)
   - HTML (*.htm)
4. Specify the name of the file attachment in the email in the Attachment name field.
5. If you want to specify a conditional file attachment name, click the ellipsis (...). The Formula editor opens.
6. Specify the formula to use to determine the name of the attachment and click OK. The Formula editor closes.
7. Click Next. The Setup [file type] export options page opens. On this page you can select additional options that are available depending on the type of file that you are creating.
8. Click Next. On Specify email settings page, specify values for the following parameters:
   - Select the format of the body text from the Format list. The available formats are Text and HTML
   - The address that the email will be sent to in the To field
   - The address that a carbon copy of the email will be sent to in the Cc field
   - The subject of the email in the Subject field
   - The body text of the email in the Message field
   - The address that will send the email in the From field
9. To specify SMTP server parameters, click SMTP settings. In the SMTP Settings window, specify whether to use a server predefined by administrator or a custom one.
   a. If you decide to use a predefined SMTP server, select the required server from the list of predefined servers.
   b. If you decide to use a custom server, specify server parameters including its address and port number. If the SMTP server requires authentication, select Use SMTP authentication, and then specify user name and password. If it is necessary to specify additional access preferences, click Advanced to open the Advanced SMTP Settings window where you can add required keywords supported by javamail. Click OK.
   
   Click OK to close the SMTP Settings window.
10. Click Finish. The Operation Wizard closes.
You must specify one or more of the following options when you export query results to a file. The options that you specify depend on the type of file that you are exporting.

### Specifying export quick report operation options

Use the export quick report operation to export quick report results to a file.

#### About this task

To specify export quick report operation options, perform the following procedure:

#### Procedure

1. Open the Specify operations page of the Action Wizard and click **Add Operation**. The Operation Wizard opens.
2. Click **Export quick report** and then click **Next**. The Specify export parameters page opens.
3. Select the file type that you want to export from the **Export type** drop-down list. Valid file types are:
   - PDF (*.pdf)
   - HTML (*.htm)
4. Specify the location to save the object. There are two options:
   a. To specify a static save location for the object, click **Path** and then click the ellipsis (...) at the bottom right of the **Destination** area. The Save As window opens.
   b. Navigate to the directory location where you want to save the object and click **Finish**. The Save As window closes.
   c. In case the **Root output directory** is specified on the **Server-Side File System** page of the Preferences window, you can just type a path and a file name or only file name in **Path** field. The file will be exported to this root directory automatically. In DataQuant for WebSphere you can specify only the path, lying in the root directory set by administrator. Otherwise, the export will be forbidden and an error will occur.
   d. To specify a conditional save location for the object, click **Formula** and then click the ellipsis (...). The Formula editor opens.
   e. Specify the formula to use to determine the location to save the object and click **OK**. The Formula editor closes.
5. Select the **Delete after task completion** check box if you want to remove the exported file from the specified location when the scheduled task is completed.
6. Select the **Mark as attachment** check box if you want to be able to attach this quick report to the email automatically.
7. Click **Next**. The Specify [file type] export options page opens. On this page you can select additional options that are available depending on the type of file that you are creating.
8. Click **Finish**. The Operation Wizard closes.

#### Related tasks:

- "Setting the directory location preference for objects generated by Export actions" on page 18

You can set preferences in DataQuant so that objects generated from various DataQuant actions are exported to a specific location on either the local file system.
or the server file system.

Related reference:

“Specifying file export options” on page 120

You must specify one or more of the following options when you export query results to a file. The options that you specify depend on the type of file that you are exporting.

Specifying export visual report to mail operation options

You use the export visual report to mail operation to send a visual report in an email.

About this task

To specify export visual report to mail operation options, perform the following procedure:

Procedure

1. Open the Specify operations page of the Action Wizard and click Add Operation. The Operation Wizard opens.
2. Click Export visual report to mail and then click Next. The Specify export parameters page opens.
3. Select the file type that you want to export from the Export type drop-down list. Valid file types are:
   - PDF (*.pdf)
   - HTML (*.htm)
4. Specify the name of the file attachment in the email in the Attachment name field.
5. If you want to specify a conditional file attachment name, click the ellipsis (...). The Formula editor opens.
6. Specify the formula to use to determine the name of the attachment and click OK. The Formula editor closes.
7. Click Next. The Setup [file type] export options page opens. On this page you can select additional options that are available depending on the type of file that you are creating.
8. Click Next. On Specify email settings page, specify values for the following parameters:
   - Select the format of the body text from the Format list. The available formats are Text and HTML
   - The address that the email will be sent to in the To field
   - The address that a carbon copy of the email will be sent to in the Cc field
   - The subject of the email in the Subject field
   - The body text of the email in the Message field
   - The address that will send the email in the From field
9. To specify SMTP server parameters, click SMTP settings. In the SMTP Settings window, specify whether to use a server predefined by administrator or a custom one.
   a. If you decide to use a predefined SMTP server, select the required server from the list of predefined servers.
   b. If you decide to use a custom server, specify server parameters including its address and port number. If the SMTP server requires authentication,
select **Use SMTP authentication**, and then specify user name and password. If it is necessary to specify additional access preferences, click **Advanced** to open the Advanced SMTP Settings window where you can add required keywords supported by `javamail`. Click **OK**.

Click **OK** to close the SMTP Settings window.

10. Click **Finish**. The Operation Wizard closes.

**Related reference:**

"Specifying file export options” on page 120

You must specify one or more of the following options when you export query results to a file. The options that you specify depend on the type of file that you are exporting.

**Specifying export visual report operation options**

Use the export visual report operation to export a visual report to a file.

**About this task**

To specify export visual report operation options, perform the following procedure:

**Procedure**

1. Open the Specify operations page of the Action Wizard and click **Add Operation**. The Operation Wizard opens.
2. Click **Export visual report** and then click **Next**. The Specify export parameters page opens.
3. Select the file type that you want to export from the **Export type** drop-down list. Valid file types are:
   - PDF (*.pdf)
   - HTML (*.htm)
4. Specify the location to save the object. There are two options:
   a. To specify a static save location for the object, click **Path** and then click the ellipsis (…) at the bottom right of the **Destination** area. The Save As window opens.
   b. Navigate to the directory location where you want to save the object and click **Finish**. The Save As window closes.
   c. In case the **Root output directory** is specified on the **Server-Side File System** page of the Preferences window, you can just type a path and a file name or only a file name in the **Path** field. The file will be exported to this root directory automatically. In DataQuant for WebSphere you can specify only the path, lying in the root directory set by administrator. Otherwise, the export will be forbidden and an error will occur.
   d. To specify a conditional save location for the object, click **Formula** and then click the ellipsis (…). The Formula editor opens.
   e. Specify the formula to use to determine the location to save the object and click **OK**. The Formula editor closes.
5. Select the **Delete after task completion** check box if you want to remove the exported file from the specified location when the scheduled task is completed.
6. Select the **Mark as attachment** check box if you want to be able to attach this visual report to the email automatically.
7. Click **Next**. The Specify [file type] export options page opens. On this page you can select additional options that are available depending on the type of file that you are creating.
8. Click **Finish**. The Operation Wizard closes.

**Related tasks:**

- **“Setting the directory location preference for objects generated by Export actions”** on page 18

You can set preferences in DataQuant so that objects generated from various DataQuant actions are exported to a specific location on either the local file system or the server file system.

**Related reference:**

- **“Specifying file export options”** on page 120

You must specify one or more of the following options when you export query results to a file. The options that you specify depend on the type of file that you are exporting.

---

**Adding a resource manipulation task**

The resource manipulation action allows you to automatically copy, paste, delete, and email resources that are generated by the scheduled task.

**About this task**

To add a resource manipulation action to a scheduled task:

**Procedure**

1. Open a task from the Task List window.
2. Click **Add Action**. The Action Wizard opens.
3. Specify a name for the action in the **Action name** field, click **Resource manipulation**, and then click **Next**. The Specify resource manipulation type page opens.
4. Click **Copy** or **Send mail** and then click **Next**. The next page of the wizard opens.
5. Specify the **Copy** or **Send email** options and click **Finish**. The Action Wizard closes and control returns to the Task window.
6. Click **OK**. The Task window closes and control returns to the Task List window.
7. Click **OK**. The Task List window closes.

**Specifying copy resource parameters**

The copy resource manipulation action allows you to automatically copy, paste, and delete resources that are generated by the scheduled task.

**About this task**

To specify copy resource parameters:

**Procedure**

1. Open the Specify resource manipulation type page of the Action Wizard.
2. Click **Copy** and then click **Next**. The Specify copy resource parameters page opens.
3. Specify the resource that you want to copy. There are two options:
   a. To specify the static source location of the object, click **Path** and then click the ellipsis (…) at the bottom right of the **Source** area. The Open window opens.
b. Navigate to and select the object and click **Finish**. The Open window closes.

c. To specify a conditional source location for the object click **Formula** and then click ellipsis (...). The Formula editor opens.

d. Specify the formula to use to determine the location of the object and click **OK**. The Formula editor closes.

4. Specify where you want to save the resource. There are two options:

   a. To specify the static source destination of the resource, click **Path** and then click the ellipsis (...) at the bottom right of the **Source** area. The Save As window opens.

   b. Navigate to the location where you want to save the resource and its file name and click **Finish**. The Save As window closes.

   c. In case the **Root output directory** is specified on the **Server-Side File System** page of the Preferences window, you can just type a path and a file name or only a file name in the **Path** field. The file will be exported to this root directory automatically. In DataQuant for WebSphere you can specify only the path, lying in the root directory set by administrator. Otherwise, the export will be forbidden and an error will occur.

   d. To specify a conditional source destination for the resource, click **Formula** and then click the ellipsis (...). The Formula editor opens.

   e. Specify the formula to use to determine where to save the resource and click **OK**. The Formula editor closes.

5. To cut and paste the resource from the source location to the destination location, select the **Move** check box.

6. To create a new folder to contain the moved resource, select the **Create folders** check box.

7. To delete the moved copy of the resource once the task has been completed, select the **Delete after task completion** check box.

8. Select the **Mark as attachment** check box if you want to be able to attach this resource copy to the email automatically.

9. Click **Finish**. The Action Wizard closes and control returns to the Task window.

**Related tasks:**

"Setting the directory location preference for objects generated by Export actions" on page 18

You can set preferences in DataQuant so that objects generated from various DataQuant actions are exported to a specific location on either the local file system or the server file system.

### Specifying mail settings

The send mail resource manipulation action allows you to automatically send generated resources in an email.

**About this task**

To specify mail settings:

**Procedure**

1. Open the Specify resource manipulation type page of the Action Wizard.
2. Click **Send mail** and then click **Next**. The Specify mail settings page opens.
3. Specify the format of the body text in the **Format** list. The available formats are Text and HTML.
4. Specify the recipient of the email in the **To** field.
5. Specify any additional recipients in the Cc: field.
6. Specify the subject of the email in the Subject field.
7. Specify the text of the email in the Message field.
   You can attach images as well as files of other available types.
   If you want to include an embedded image in the text of the email, you must insert the following HTML snippet into the text:
   \[\text{<img src="cid:}[\text{name}]"/img>\]
   where \text{cid:} is the standard abbreviated content identifier required when referencing an embedded image within a formatted email, and where \text{name} is the name of the image file. For example, to embed an image file named \text{image1.png}, you would insert the following HTML snippet into the text of the email:
   \[\text{<img src="cid:image1.png"/>}\]
8. Specify the email address that will send the email in the From: field.
9. To specify SMTP server parameters, click SMTP settings. In the SMTP Settings window, specify whether to use a server predefined by administrator or a custom one.
   a. If you decide to use a predefined SMTP server, select the required server from the list of predefined servers.
   b. If you decide to use a custom server, specify server parameters including its address and port number. If the SMTP server requires authentication, select Use SMTP authentication, and then specify user name and password. If it is necessary to specify additional access preferences, click Advanced to open the Advanced SMTP Settings window where you can add required keywords supported by javamail. Click OK.
   Click OK to close the SMTP Settings window.
10. Click Add. The Enter path as text or formula window opens.
11. Specify the path to the resource that you want the task to send in an email and click OK. The Enter path as text or formula window closes.
12. In the Attachments area, select the required options from two lists.
   a. From the first list select the attachment that you want to add automatically. The available options are:
      • No automatic attachments not to attach any resource automatically.
      • Attach marked only to attach only the resources that were marked in the export or copy operations.
      • Attach all to attach all resources created in the export or copy operations.
   b. From the second list select the procedure that you want to apply to the attachments after the action execution. The available options are:
      • Clear all automatic attachments to delete all attachments sent automatically after the action execution.
      • Clear automatic attachments in the group to delete all automatic attachments, which are created for a group of actions with the sending an email action, after the action execution.
      • Do not clear automatic attachments to store all automatic attachments after the action execution.
13. Click Finish. The Action Wizard closes and control returns to the Task window.
Adding an action group

You can use action groups to create a series of actions that should be performed successively.

Before you begin

Before creating an action group, you must first create a scheduled task.

About this task

To add an action group to a scheduled task:

Procedure

1. Open the Task window in one of the following ways:
   - Double-click a task in the Tasks list;
   - Select a task from the Tasks list, and then click Edit Task.
2. In the Task window, click Add Action to open the Action wizard.
3. Type a name in the Action name field.
4. Select Action Group, and then click Finish.

Adding actions to an action group

You can use drag-and-drop method to add actions to an action group.

Before you begin

You must create an action group.

You must create an action.

About this task

To add an action to an action group:

Procedure

1. Open the Task window in one of the following ways:
   - Double-click a task in the Tasks list;
   - Select a task from the Tasks list, and then click Edit Task.
2. In the Task window, select an action that you want to add to a group.
3. Drag the selected action onto an action group item.

What to do next

Next: Setting Iterator parameters
Related tasks:
“Adding an execute object action” on page 139
The execute object action allows you to automatically execute queries, a visual report, procedure, or quick report, and export, mail, or print it.
“Adding a resource manipulation task” on page 148
The resource manipulation action allows you to automatically copy, paste, delete, and email resources that are generated by the scheduled task.
Removing actions from an action group

You can use drag-and-drop method to remove actions from an action group.

Before you begin

You must create an action group and populated it with actions.

About this task

To remove an action from an action group:

Procedure

1. Open the Task window in one of the following ways:
   - Double-click a task in the Tasks list;
   - Select a task from the Tasks list, and then click Edit Task.
2. In the Task window, expand an action group that you want to edit.
3. Select an action that you want to remove from the action group.
4. Drag the selected action outside the action group.

Setting Iterator parameters

You can use Iterator to run an action group for several times in a cycle. You use the Iterator Parameters window to manage Iterator preferences.

Before you begin

You must create an action group.

About this task

To set Iterator parameters:

Procedure

1. In the Actions list of the Task window, right-click the action group that you want to run in a cycle, and then select Set Iterator from the pop-up menu.
2. In the Iterator Parameters window, click the ellipsis (...) to specify a query for Iterator.
   The columns of this query define Iterator variables and the number of rows specifies the number of iterations.
3. In the Open window navigate to and select an object that you want to use as a query for Iterator.
   The following objects can be used as a query:
   - Charts
   - Queries
   - Visual Queries
   - Analytical Queries
   - Dynamarts
   - Compound Dynamarts
   - Tables
4. Click Finish. The path to the selected object is displayed in the Query path field.
5. The data source that contains the selected object is added to the Data Sources list. You can specify your login information for a data source in the Login and Password fields.

6. To add another data source to the list, perform the following procedure:
   a. Click Add Data Source to open the Add Data Source window.
   b. From the Data Source list, select a data source that you want to add.
   c. Type your login information in the User Name and Password fields.
   d. Select the Remember the password check box if you want DataQuant to remember the password for the user ID that you entered in the User Name field.

7. To remove a previously added data source from the list, select this data source, and then click Remove Data Source.

8. In the Variables list, link the query columns with the prompts used in the source object that you specify in Execute object actions. Click in the Prompt cell and select the prompt from the drop-down list.

9. Use the Continue on failure check box to specify whether to continue Iterator work when an error occurs. If the check box is selected and an error occurs within any iteration, Iterator skips this iteration and continues working. If an error occurs and the check box is cleared, Iterator stops working.

   Note: All error messages can be viewed in the Output view.

10. Click OK to close the Iterator Parameters window.

What to do next

Next: Managing prompts in an action group

Related tasks:

“Adding an execute object action” on page 139

The execute object action allows you to automatically execute queries, a visual report, procedure, or quick report, and export, mail, or print it.

Managing prompts in an action group

You can create a scheduled task that runs a variety of data objects such as queries, quick reports, visual reports, and procedures containing parameters.

About this task

You use the Manage Shared Prompts window to define how the parameter values must be passed to the source objects specified in the task.

Note: Depending on the type of an object, parameters are represented by different elements. For queries, parameters are prompts defined for these queries. For quick reports and visual reports, parameters are prompts specified in queries on which these reports are based. For procedures, parameters are variables defined in these procedures.

You can group similar prompts used in the source objects so that all the prompts within one prompt group receive the value passed to this group. By default, there are as many groups as prompts in the source objects within the action group. Each group stores one particular prompt and borrows its name. You can move prompts between the groups. If you move the last prompt from the group, this group is removed automatically because empty groups are not allowed.
You must create an action group and populated it with execute object actions.

To manage shared prompts:

**Procedure**
1. Open the Task window by selecting a task from the **Tasks** list and clicking **Edit Task**.
2. In the Task window, right-click an action group, and then select **Manage Prompts**.
3. From the **Available prompts** list in the Manage Shared Prompts window, select a prompt that you want to add to a group.
4. From the **Shared prompts** list, select a group that you want to add the prompt to.
5. Click **Add to Group**. The selected prompt is added to the prompt group.
6. To remove a prompt from a prompt group, select a prompt from the **Shared prompts** list, and then click **Detach Prompt**.
7. To rename a prompt group, select the prompt group that you want to rename, and then click **Rename prompt group**.
8. In the Prompt Group window, specify the name and string to display, and then click **OK**.
9. To change the order of the prompt groups in the **Shared prompts** list, select the node of the prompt group or a prompt in the group, and click **Move Prompt Group Up** or **Move Prompt Group Down**.
10. Add an Iterator query for the action group and link the query columns with the parameters used in the source objects that you specified in actions within the action group.

**Related tasks:**

“Working with prompts in queries” on page 50

You can filter your queries or dynamarts by adding prompts that direct the query or dynamart to focus on specific values or ranges. This limits the amount of data to be queried and provides more focused query results. The filtering works on the database level.

---

**Specifying variables for actions and action groups**

You can create a scheduled task that runs a variety of data objects such as queries, quick reports, visual reports, and procedures containing parameters.

**About this task**

You use the Set Variables window to define how the parameter values must be passed to the source objects which are specified in the task. You can also use the specified parameters in actions themselves as substitution variables. For example, you can form the file system path by using parameters.

**Note:** Depending on the type of an object, parameters are represented by different elements. For queries, parameters are prompts defined for these queries. For quick reports and visual reports, parameters are prompts specified in queries on which these reports are based. For procedures, parameters are variables defined in these procedures.
You can set parameter values for both action groups and single actions. You can also set the iterator for the same group. The parameters provided by the single action take precedence, then goes parameter values set for the group, then parameter values set by the iterator.

To specify parameter values:

**Procedure**

1. Open the Task window by selecting a task from the Tasks list and clicking **Edit Task**.
2. In the Task window, click the **Set Variables** toolbar button. All the parameters specified for the selected data object are displayed in the list of parameters in the Set Variables window.

   **Note:** If you decide to set parameter values to an action group to which you have already assigned shared prompts, the specified shared prompts are displayed instead of parameters defined in the source object.

3. To add a parameter, click **Add Variable**, and then specify its name and the value in the Add Variable window.
4. To edit or remove a variable, select the variable you want to edit or remove, and click **Edit** or **Remove**.
5. To set the parameter value, open the Add Variable or Edit Variable window and enter either the constant value or a formula in the **Value** field. To open the Formula editor window, click the ellipsis (...) next to the **Value** field.

   **Note:** You can also set the parameter value directly in the Set Variables window by simply typing it in the required cell of parameters table.

6. You can change the order in which parameters are calculated by clicking **Move Up** and **Move Down**. Parameters are calculated sequentially as they appear in the list and those that have already been calculated can be used to calculate other parameters.

7. Click **OK** to close the Set Variables window.

---

**Scheduling local tasks**

Use the specific to your operating system scheduler window or application schedule task window to schedule a task to run.

**Before you begin**

You must have a task to schedule.

**About this task**

If you select **Native** on the **General** tab in the Preferences window, DataQuant integrates with the built-in scheduler of the operating system to schedule tasks. For more information about the built-in scheduler that is available, check your operating system documentation, or use the context help in the scheduler window.

If you select **DataQuant for Workstation scheduler** on the **General** tab in the Preferences window, the application schedule task window opens.

To schedule a task with DataQuant for Workstation scheduler:
Procedure

1. Select View > Scheduled Tasks. The Task List window opens.
2. On the Local tasks tab, select the task that you want to schedule and click Schedule Task.
3. Specify the operating system user name and password in the User and Password fields.
4. Optional: In the Working directory field, specify the directory for temporary files, which are created during the task execution.
   - Select the Delete when done check box to delete the task from the scheduler window after its execution.
   - Select the Run only if logged on check box to run the task only if the user with the credentials that were specified in the User and Password fields is logged on the operating system.
   - Select the Disabled check box to disable the task execution.
5. Specify the schedules that you want to apply to the task in the Schedules list.
6. To add a schedule, click Add Schedule.
7. Optional: To remove a schedule, which you do not need, click Remove Schedule.
8. Select the execution frequency of the task from the Type of schedule list.
9. Specify the frequency details in the area next to the Type of schedule list. The options vary depending on the selected type of schedule.
10. Click OK to close the scheduler window.

Sending HTML tables in email

You can send query results as HTML tables in the email when you work with scheduled tasks.

Before you begin

Make sure that the selected format of a letter is HTML and the table you want to add is saved in the HTML format too.

About this task

The inserted tables appear in the body of the letter in HTML format.

Adding an HTML table is available only for the following tasks of the scheduler:
- Exporting query to mail task. As a result of this task you get query result set in HTML format, which can be sent in the email.
- Sending mail resource manipulation task. In this task you can insert the necessary HTML tables created when performing other tasks.

To insert an HTML table in the body of an HTML letter:

Procedure

1. In the Action Wizard window, click an ellipsis button (...) in the Message field. The Formula editor opens.
2. From the list of the functions expand the Scheduler and select insertHTMLTable or type the name of this function in the Filter field.
3. Double-click the function and specify its parameters.
4. Click **OK** to add the function. The specified table is added to the letter.

**Example**

You can add `grid.html` table to the letter with the following parameters:

- When you work with export task, just add the function without any parameters
  ```javascript
  $insertHTMLTable()
  ```
- When you work with resource manipulation task, you specify only one parameter, in case the exported HTML file has UTF-8 encoding.
  ```javascript
  $insertHTMLTable("grid.html")
  ```
- You must specify both parameters if the encoding of the exported HTML file differs from UTF-8.
  ```javascript
  $insertHTMLTable("grid.html", "UTF-16")
  ```
Chapter 10. Generating reports

Report developers can select from one of several formats that can be used to generate a report for a set of query results.

The available reporting formats are:

• Text-based, tabular classic reports that are backward-compatible with the legacy QMF infrastructure.
• Graphical visual reports authored using a built-in visual designer.

Note: You can navigate a classic report using the arrow, Home, and End keys on the keyboard. The Up and Down arrows scroll through the pages of the report vertically. The Left and Right arrows scroll through a page horizontally. The Home key navigates back to the top of the first page of the report. The End key navigates to the end of the last page of the report.

Creating a new report

You can create a new report using the Display Report wizard.

About this task

To generate a report in any of the formats:

Procedure

2. From the first page of the wizard, select whether you will be creating a new report or using an existing report. Your choices are:
   • Create a new report
   • Use an existing report stored in a file
   • Use an existing report stored in the QMF catalog
   • Use an existing report stored in the repository
3. Click Next.
4. Depending on your selection one of the following dialogs opens:
   • If you selected Create a new report, the second page of the wizard requests that you specify what type of report you will create. The available reporting options are:
     – Create a classic report to create a classic report.
     – Create a Visual report to create a visual report.
     For each type of report you must specify whether the report will be generated based on how the current query is formatted or on simple default formatting. Select Create from query to specify that the report will be generated based on how the current query results are formatted. Select Default to specify that the report will be generated on simple default formatting. For classic report you must specify the Fetching options. Select Use available data to include in the report only the data that has already been fetched. Select Fetch all data to complete the process of fetching the data before the report is generated so that all data is included.
• If you selected **Use an existing report stored in a file**, the second page of the wizard requests that you specify the path to the file that contains the report in the **Name** field.

• If you selected **Use an existing report stored in the QMF catalog**, the second page of the wizard requests that you specify the database where the QMF catalog resides in the **Database** field; and the report owner, name and type in their respective fields. To see a list of reports click **List Objects**.

• If you selected **Use an existing report stored in a repository**, the second page of the wizard requests that you select from the repository or a repository workspace the report that you want to open.

5. Click **Finish**. The report is displayed in the active editor window.

---

### Selecting a classic report's data source object

You can select a new data source object for the current classic report or specify that a form will always use a specific data source object to produce a classic report.

#### About this task

Changing data source objects by using the Data Source Object dialog makes it easy to view different sets of query results data in the same classic report format. DataQuant will run the newly selected data source object, format the query results using the current form, and display the updated report in the active editor window.

Specifying that a particular data source object will always be used when a form is opened streamlines the steps required to create a classic report. If specified, when DataQuant opens such form, it will find the assigned data source object, run the object, format the query results according to the specifications of the form and display the generated classic report in the editor window. If you do not specifically assign a data source object to a form, then when you open that form, DataQuant formats the currently active query results into a report. If you want specific query result data for the report, you must find the object, run the object and then generate the report.

To specify a data source object:

#### Procedure

1. Open the Data Source Object window.
   
   With a form in the active window, select **Form > Data Source Object**.

2. Specify where the data source object is located by selecting one of the following options:
   
   - Click **From file** to specify that the data source object is saved to a file located on a local or network drive. The required fields become available. Specify the name of the file in the **Name** field. You can search the list of available files by clicking the **Browse** button.
   
   - Click **From data source** to specify a data source object that is stored on a database. The required fields become available. Specify the data source where the object is stored in the **Data Source** field. Specify the owner, name, and type for the data source object in the **Owner**, **Name**, and **Type** fields. You can click the **List Objects** button to search the database for an object.
• Click From repository to specify a data source object that is saved in a repository. The required fields become available. Expand repository directories and folders to expose the data source object that you want to select.

• Click From open object to get the data source object from one of the currently open query objects. Select the name of the object from the Title list box of the open query objects.

3. If the data source object that you have selected produces multiple result sets, as in the case with stored procedures or multiple SQL statement queries, specify the index number of the result set that you want to use in the Result set field.

4. Check the Always use this object when opening this document check box to specify that whenever the currently active form is opened, the object that you specified will be used as the data source object. This option is not available when the selected data source object is a currently open object.

5. Click OK to generate the report using the selected data source object. If you specified that the currently selected data source object always be used when this form is opened, the next time you open the form the object will be run, and the query results will be formatted according to the specifications of the form into a report.

Listing data source objects

You use the Object list window to list the objects that have been saved to the data source.

About this task

You can select an object from this list to be used as the data source for a report.

To list the objects on a data source:

Procedure

1. Open the Data Source Object window.
   With a form in the active window, select Form > Data Source Object.
2. Select From data source radio button and then click List Objects.
3. From the Data Source drop-down list, select the name of the data source whose objects you would like to list.
4. In the Owner field, enter the owner of the objects that you want to list. You must specify the owner in the correct format for identifiers.
5. In the Name field, enter the name of the object that you want to list. You must specify the name in the correct format for identifiers. You can specify a matching pattern to match multiple names.
6. Click Refresh List to initiate a search for the objects on the database that match the search criteria that you specified in the Owner and Name fields. Only those objects that you are authorized to use are included in the list. The results are returned in the Object List table.
7. From this list, select the object that you want to use as the classic report data source object.
8. Click OK. The Object List window closes. Control returns to the Data Source Object window.
Exporting reports to a file

You can export a classic or quick report to a file in text, HTML or PDF format. If your report is a classic one, you can export it as a text, HTML or PDF file. If your report is a quick one, you can export it as an HTML or PDF file.

About this task

To export a report:

Procedure

1. Open the Export Report wizard in one of the following ways:
   • With a report in the active window, select File > Export to open the Select page of the Export wizard, and then select Report from the list of available export wizards.
   • With a report in the active window, click the Export toolbar button.
2. Select the report that you want to export from the Reports list box.
3. If you are using DataQuant for Workstation, specify where the report file should be saved in the Report file field. You can use the Browse button to search for the location.
4. Select the type of file that you are exporting. You can export a classic report as a text file (*.txt), an HTML file (*.htm;*.html), or a PDF file (*.pdf). You can export a quick report as an HTML file (*.htm;*.html) or a PDF file (*.pdf).
5. If you are exporting a PDF file, click the Next button. The Set up PDF export options page opens. Select the Add Document Information check box and specify character strings in the Title, Author, Subject and Keywords fields to add document information to the PDF. Select the Use only ANSI fonts check box to use only ANSI fonts in the PDF.
6. Click Finish.
   • If you are using DataQuant for Workstation, the report is exported to a file in the format that you selected.
   • If you are using DataQuant for WebSphere, a Save window opens that list the individual files for download, and an archived ZIP file (*.zip) that contains all of the files for download. Click one or more of the hyper-links to download the files. When you have finished downloading the files, click OK.

Creating classic report forms

Classic reports are generated using query results as the data source and a form template. The form template, referred to as a "form", specifies how you want the query results formatted in the report.

About this task

Forms are considered objects and they can be saved in your repository, in the QMF catalog, or in a file. When you open a form object that has been saved, you are actually running the form object to generate the report. When opened, forms automatically use the currently active query results as the data source.

Alternatively, you can specify that a specific set of query results data always be used with a specific form to create a report. If you select this option, when the
form object is opened a specific query object is run to obtain the requested query results and then those query results are formatted into a report using the form specifications.


**Designing a classic report form**

You can create and edit forms using the Design page of the Classic Report editor window. Generated reports are displayed in the Report page.

**About this task**

To create a form:

**Procedure**

1. Open the Design page for a classic report in one of the following ways:
   - Select File > New > Form. The Create New Form wizard opens. Specify the name of the form in the Name field and where the form will be saved in the Data source field. Click Finish.
   - Select the New Form toolbar button. Click the Design tab to open the Design page of the Form window.

2. The Design page is divided into two sections. To the left is the Form structure tree. Each node in the tree represents a section of the report. As you select a node from the tree, the fields that you have to specify for that section of the report are displayed.

3. To design your form, select one or more sections from the Form structure tree and enter the corresponding fields. A classic report includes the following sections:
   - Main
   - Breaks
   - Calculations
   - Columns
   - Conditions
   - Details
   - Final
   - Options
   - Page

4. When you have completed designing the form, you can save the form in your repository, in the QMF catalog, or to a file.

**Specifying Main formatting options**

The formatting fields contained within the Main section of a report provide the formatting instructions for the most common elements of a report including columns, form headings, footings, and breaks.

**About this task**

Some of the elements that you define in the Main section of the report can also be defined in other sections.
### Table 17. Main section elements for a classic report

<table>
<thead>
<tr>
<th>Element</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total width of report columns</td>
<td>This property displays the total width of the tabular data in the report.</td>
</tr>
<tr>
<td>Column properties</td>
<td>In the table at the top of the page, you can specify which columns to display and set their properties. You can use the buttons above the table to add or remove columns, open Edit column dialog, and change the columns order. See <a href="#">Column properties</a> for details.</td>
</tr>
<tr>
<td>Page</td>
<td>Use the Heading text and Footing text fields to specify the heading and footing of the page.</td>
</tr>
<tr>
<td>Final</td>
<td>Use the Text field to specify the final line text.</td>
</tr>
<tr>
<td>Breaks</td>
<td>Use the following elements to specify breaks properties:</td>
</tr>
<tr>
<td></td>
<td>• <strong>New page for break</strong></td>
</tr>
<tr>
<td></td>
<td>Check this check box to start a new page in the report when the current break level occurs.</td>
</tr>
<tr>
<td></td>
<td>• <strong>Footing text</strong></td>
</tr>
<tr>
<td></td>
<td>Use this field to specify the break footing text.</td>
</tr>
<tr>
<td>Outlining for break columns</td>
<td>Break columns are the columns with the Usage code set to Break. The report lines are grouped by the value in a break column. If you check this check box the value is displayed only in the first line of a group. Otherwise, the value is displayed in every line of a group.</td>
</tr>
<tr>
<td>Default break text (*)</td>
<td>Check this check box to set asterisk as the default break text. If there is no aggregation set for any of the columns, an asterisk is shown at the end of the break line.</td>
</tr>
</tbody>
</table>

### Column properties

You can edit the column properties in the table or by using the Edit column dialog. Use the following fields to specify column properties.

- **Num:**
  Use this property to identify the column. This number is used to resolve form variables, and to determine how query result columns map to report columns. Enumeration starts at 1. You can change the column number by moving it up or down in the columns list. For this, use the Move Up Column and the Move Down Column buttons.

- **Heading:**
  Use this field to specify the column heading, up to 40 characters in length. Use the underscore character ( _ ) to create a multiline heading. Leading underscores are ignored. For example, **FIRSTSECOND** is displayed as:
  
  \[
  \text{FIRST}
  \text{SECOND}
  \]

- **Usage or Usage code:**
  Use this field to assign a usage code to the column. Usage codes provide summary information about the data in a column. For example, usage codes can provide total summary information at the end of a column, or partial summaries at control breaks in a table. The usage codes available depend on the data in the
column and the type of summary. From the drop down list of available usage codes, select the appropriate one. Refer to the online help system for more information on each usage code.

- **Indent:**
  Specify the number of blank spaces to the left of the column. The range is between 0 and 999.

- **Width:**
  Use this field to specify the width of the column. The number can be between 1 and 32,767. Make sure you allow enough width for currency symbols, the negative sign, and other special characters. If the column is too narrow for numeric data, it is replaced with asterisks. If the column is too narrow for character or date/time data, it is truncated based on the alignment specified. You can use a wrapping edit code to have the data wrap within the column width.

- **Edit:**
  Select an edit code to control how data is formatted in a report. Edit codes are available for character, date, graphic, numeric, time, timestamp data. Also, user-defined edit codes can be created.
  From the drop down list of available edit codes, select the appropriate one. Refer to the online help for more information about edit codes.

- **Seq:**
  Use this field to specify the sequence of the columns in the report. If two columns have the same sequence number, they appear in the report in the column number order.

  **Note:** The **Seq** field is ignored in reports that use the ACROSS usage code or if the Automatic reordering of report columns is selected on the Form Options window.

**Specifying Breaks formatting options**
The formatting elements contained within the **Breaks** section of a report specify the characteristics, content, and placement of up to six subtotal lines in a report, along with break heading text and break footing text.

**About this task**

*Table 18. Breaks section elements for a classic report*

<table>
<thead>
<tr>
<th>Element</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>New page for break</td>
<td>Check this check box to start a new page in the report when the current break level occurs.</td>
</tr>
<tr>
<td>Repeat detail heading</td>
<td>Check this check box to repeat the current detail heading with the current break level.</td>
</tr>
<tr>
<td>Blank lines before heading</td>
<td>Use this field to specify the number of blank lines to be included before the first break heading line. This value must be a number between 0 and 999.</td>
</tr>
<tr>
<td>Blank lines after heading</td>
<td>Use this field to specify the number of blank lines that are to be included after the last break heading line. This value must be a number between 0 and 999.</td>
</tr>
</tbody>
</table>

Chapter 10. Generating reports  165
Table 18. Breaks section elements for a classic report  (continued)

<table>
<thead>
<tr>
<th>Element</th>
<th>Description</th>
</tr>
</thead>
</table>
| Break heading text          | You use the buttons above the Break heading text table to add, edit, or remove lines, and also change the lines order. Use the following fields to specify the break heading text:  
  - **Line**: Use this field to identify lines of break heading text and their relative positions to each other in the report. You can specify any number from 1 to 999 or a blank. You do not have to start with line one and you do not have to specify consecutive line numbers. If you use the same line value for more than one line, those lines are joined according to the value specified in the Alignment field. Lines with the same value in the Line field overlay each other if they are longer than the report width, or if their values in the Alignment field conflict.  
  - **Alignment**: Use this field to specify where horizontally on the line this text is placed. Valid alignment values are LEFT, RIGHT, CENTER, APPEND, or the number of a specific position in the line. If you specify APPEND, the text is placed immediately following the previous line of the text.  
  - **Text**: Use this field to specify the text that appears in the break heading. You can use form variables and global variables in this field. |
| New page for footing        | Check this check box to start a new page in the report when the current break level ends.                                                                                                                    |
| Put break summary at line   | Use this field to specify where vertically in the break footing lines, the break summary line is placed. This value must be a number between 1 and 999. If you do not want a summary line, specify NONE.                                      |
| Blank lines before footing  | Use this field to specify the number of blank lines to be included before the first break footing line. This value must be a number between 0 and 999, or BOTTOM which is treated as the value 0 when producing reports.                        |
| Blank lines after footing   | Use this field to specify the number of blank lines that are to be included after the last break footing line. This value must be a number between 0 and 999.                                                          |
Table 18. Breaks section elements for a classic report (continued)

<table>
<thead>
<tr>
<th>Element</th>
<th>Description</th>
</tr>
</thead>
</table>
| Break footing text | You use the buttons above the Break footing text table to add, edit, or remove lines, and also change the lines order.  
Use the following fields to specify the break footing text:  
- **Line**: Use this field to identify lines of break footing text and their relative positions to each other in the report. You can specify any number from 1 to 999 or a blank. You do not have to start with line one and you do not have to specify consecutive line numbers. If you use the same line value for more than one line, those lines are joined according to the value specified in the Alignment field. Lines with the same value in the Line field overlay each other if they are longer than the report width, or if their values in the Alignment field conflict.  
- **Alignment**: Use this field to specify where horizontally on the line this text is placed. Valid alignment values are LEFT, RIGHT, CENTER, APPEND, or the number of a specific position in the line. If you specify APPEND, the text is placed immediately following the previous line of the text.  
- **Text**: Use this field to specify the text that appears in the break footing. You can use form variables, HTML variables, and global variables in this field.|

Using HTML variable within the Text field:

You can insert HTML variables in Text fields in order to format the text data for HTML presentation. You can place these HTML variables in any section of a form.

About this task

Table 19. HTML variables

<table>
<thead>
<tr>
<th>HTML Variable</th>
<th>Use this variable to...</th>
</tr>
</thead>
<tbody>
<tr>
<td>IMAGEn</td>
<td>Places the contents of column ‘n’ inside an HTML image reference tag, making graphic files visible in reports.</td>
</tr>
<tr>
<td>LINKn</td>
<td>Places the contents of column ‘n’ inside an HTML link tag, which is used to add links to other HTML pages.</td>
</tr>
<tr>
<td>MAILTOn</td>
<td>Places the contents of column ‘n’ inside an HTML mail to tag for an email link.</td>
</tr>
<tr>
<td>ANCHORn</td>
<td>Places an anchor point at the location of the variable value. These anchors are referenced from within the report, such as the top of the page or the beginning of a break section.</td>
</tr>
<tr>
<td>RFn</td>
<td>Places the text contained in the DSQQW_HTML_REFTXT (a global variable, which contains text that appears in a report; the default = 'BACK TO') into an HTML reference tag, and is used to create links to anchors created with the ANCHORn variable in the report.</td>
</tr>
</tbody>
</table>
**Specifying Calculations formatting options**

You use the **Calculations** options to define report calculation expressions.

**About this task**

You can specify the Calculations formatting options in the **Calculations** table. Use the buttons on the interface to add, edit or remove Calculations, and also change the Calculations order.

Use the elements described in the following table to set Calculation properties.

*Table 20. Calculations option elements for a classic report*

<table>
<thead>
<tr>
<th>Element</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ID</td>
<td>Specify a number, between 1 and 999, that identifies the calculation expression.</td>
</tr>
<tr>
<td>Expression</td>
<td>Type a valid expression, up to 50 characters. Form variables cannot be used in this field.</td>
</tr>
<tr>
<td>Pass nulls?</td>
<td>Specify whether to pass the expression for evaluation when a variable in the definition has a null value.</td>
</tr>
</tbody>
</table>
| Width | Specify the width used to format the result of the calculated expression when used as a variable in the form.  
Click in the column and specify the needed value by using the spin button. |
| Edit | Specify the edit code, which is used to format the result of this calculated expression when used as a variable in the form.  
Click in the column to open a drop down list of available edit codes and select the needed one.  
See the online help for more information about edit codes. |

**Specifying Columns formatting options**

You use the **Columns** formatting options to control the appearance and formatting of columns in the report.

**About this task**

You can specify the Columns formatting options in the table in the **Columns** section of the report. You use the buttons above the table to add or remove columns, open Edit column dialog, and change the columns order.

You can edit the column characteristics in the table or by using the Edit column dialog.

Definable characteristics include column heading, column usage, indentation, width, edit code, and sequence. The **Total width of report columns** property displays the total width of the tabular data in the report.
Columns formatting options that you define in the Columns section of the report can also be defined in the Main section.

*Table 21. Columns option elements for a classic report*

<table>
<thead>
<tr>
<th>Element</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Num</td>
<td>Specifies the number that identifies the column. This property is used to resolve variable references, and to determine how query result columns map to report columns. Enumeration starts from 1. You can change the column number by moving it up or down in the columns list. For this, use the Move Up Column and the Move Down Column buttons.</td>
</tr>
<tr>
<td>Heading</td>
<td>Use this field to specify the column heading, up to 40 characters in length. Use the underscore character to create a multiline heading. Leading underscores are ignored. For example, FIRST_SECOND is displayed as: FIRST SECOND</td>
</tr>
<tr>
<td>Usage</td>
<td>Specify a usage code that will be used to collect summary information about the data in a column, such as total summary information at the end of a column, or partial summaries at control breaks in a table. The usage codes that are available depend on the data in the column and the type of summary. See the online help for more information about usage codes.</td>
</tr>
<tr>
<td>Indent</td>
<td>Specifies the number of blank spaces to indent from the left of the column. The range is between 0 and 999.</td>
</tr>
<tr>
<td>Width</td>
<td>Specify the width of the column. The range is between 1 and 32767. If the column is too narrow for numeric data, the data is replaced with asterisks. If the column is too narrow for character or date/time data, the data is truncated (based on the alignment specified). To avoid this, you can use a wrapping edit code to have the data wrap within the column width. See the online help for more information about edit codes.</td>
</tr>
<tr>
<td>Edit</td>
<td>Specify an edit code, which controls how data is formatted in a report. Edit codes are available for character, date, graphic, numeric, time, timestamp data. Also, user-defined edit codes can be created. See the online help for more information on edit codes.</td>
</tr>
<tr>
<td>Seq</td>
<td>Specifies the sequence number for the columns in the report. If two columns have the same sequence number, they appear in the report in column number order, in the Num field. <strong>Note:</strong> The sequence number is ignored in reports using the ACROSS usage code or if you check the <strong>Automatic reordering of report columns</strong> check box.</td>
</tr>
</tbody>
</table>

**Specifying Conditions formatting options**

You use the Conditions formatting options to define conditional formatting constraints, such as suppressing records that do not meet certain characteristics.
About this task

You can specify the Conditions formatting options in the **Conditions** table. You use the buttons on the interface to add, edit, or remove condition lines, and change the condition lines order.

Use the elements described in the following table to set condition properties.

**Table 22. Conditions option elements for a classic report**

<table>
<thead>
<tr>
<th>Element</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ID</td>
<td>Specify a number, between 1 and 999, that identifies the conditional expression.</td>
</tr>
<tr>
<td>Expression</td>
<td>Type a valid expression, up to 50 characters. An expression that evaluates to 1 is considered true; anything else is considered false. Form variables cannot be used in this field.</td>
</tr>
<tr>
<td>Pass nulls</td>
<td>Specify whether to pass the expression for evaluation when a variable in the definition has a null value.</td>
</tr>
</tbody>
</table>

**Specifying Details formatting options**

You use the **Details** formatting fields to define report detail headings and body text.

About this task

This is where you can combine or replace tabular data with free-form text to create form letters or address labels. Each detail variation of headings and text can correspond to a condition specified using the **Conditions** node.

When you choose the **Details** node in the Form structure tree, the **Add Detail** and **Remove Detail** buttons appear above the tree. By using these buttons you can add or remove Details.

**Table 23. Detail section elements for a classic report**

<table>
<thead>
<tr>
<th>Element</th>
<th>Description</th>
</tr>
</thead>
</table>
| Enable                                       | Specify when to enable the detail variation:  
  • **YES** indicates that the variation is always selected for formatting.  
  • **NO** indicates that the variation is never selected for formatting.  
  You can conditionally enable the variation by referring to a Form Conditions expression using the Cnn and Enn specifications. |
| Include column headings with detail headings | Check this check box to display column headings with detail headings.                                                                                                                                 |
Table 23. Detail section elements for a classic report (continued)

<table>
<thead>
<tr>
<th>Element</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Detail heading text</td>
<td>You use the buttons above the Detail heading text table to add, edit, or remove lines, and also change the lines order. Specify the lines of detail heading text:</td>
</tr>
<tr>
<td></td>
<td>• <strong>Line</strong>: specify where the text appears vertically in the detail heading. You do not have to start with line 1 and you do not have to specify consecutive line numbers.</td>
</tr>
<tr>
<td></td>
<td>• <strong>Alignment</strong>: specify where the text is placed horizontally on the line. Valid alignment values are LEFT, RIGHT, CENTER, APPEND, or the number of a specific position in the line. If you specify APPEND, the text is placed immediately following the previous line of text.</td>
</tr>
<tr>
<td></td>
<td>• <strong>Text</strong>: type the text that appears in the detail heading. You can use form variables, global variables, and HTML variables in this field.</td>
</tr>
<tr>
<td>New page for detail block</td>
<td>Check this check box to start a new page in the report for the detail block.</td>
</tr>
<tr>
<td>Blank lines after block</td>
<td>Specify the number of blank lines after the last detail block line. The range is between 0 and 999.</td>
</tr>
<tr>
<td>Repeat detail heading</td>
<td>Check this check box to repeat the detail heading before each occurrence of the detail block.</td>
</tr>
<tr>
<td>Put tabular data at line</td>
<td>Specify where the tabular data line is placed vertically in the detail block. The range is between 1 and 999. If you do not want a tabular data line, specify NONE.</td>
</tr>
<tr>
<td>Keep block on page</td>
<td>Check this check box to keep the detail block on one page. If you check this check box and a detail block is too long to fit on one page, the detail block is started on a new page. If you do not check this check box, detail blocks can split across two or more pages.</td>
</tr>
<tr>
<td>Detail block text</td>
<td>You use the buttons above the Detail block text table to add, edit, or remove lines, and also change the lines order. Specify the lines of detail block text:</td>
</tr>
<tr>
<td></td>
<td>• <strong>Line</strong>: this number specifies where the text is placed vertically in the detail block. You do not have to start with line 1 and you do not have to specify consecutive line numbers.</td>
</tr>
<tr>
<td></td>
<td>• <strong>Alignment</strong>: specify where the text is placed horizontally on the line. Valid alignment values are LEFT, RIGHT, CENTER, APPEND, or the number of a specific position in the line. If you specify APPEND, the text is placed immediately following the previous line of text.</td>
</tr>
<tr>
<td></td>
<td>• <strong>Text</strong>: type the text that appears in the detail block. You can use form variables, global variables and HTML variables in this field.</td>
</tr>
</tbody>
</table>
Specifying Options formatting options
You use the Options formatting options to define miscellaneous options that control the appearance of the report.

About this task

Table 24. Options section elements for a classic report

<table>
<thead>
<tr>
<th>Element</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Detail spacing</td>
<td>Specify the number of lines for the spacing between tabular data lines or detail blocks. This value must be a number between 1 and 999.</td>
</tr>
<tr>
<td>Line wrapping width</td>
<td>This field applies only to forms created using the ISPF version of the application QMF for TSO/CICS. Specify the number of characters at which to wrap the columns in the report. The range is between 0 and 999 or the word NONE.</td>
</tr>
<tr>
<td>Report text line width</td>
<td>For reports, specify the width of the final text, detail heading text, detail block text, and break text in a report. The range is between 1 and 32767, or the words DEFAULT or COLUMNS. For DEFAULT, break footing text and final footing text use the full width of all columns up to the first summary column. For COLUMNS, all text areas use the full width of all columns.</td>
</tr>
<tr>
<td>Number of fixed columns in report</td>
<td>Specify the number of columns that remain in place when you scroll reports horizontally on the screen or break reports onto multiple pages when printing. The range is between 1 and 999, or the word NONE.</td>
</tr>
<tr>
<td>Outlining for break columns</td>
<td>Select this check box to display the value in columns with the BREAK usage code when the value changes.</td>
</tr>
<tr>
<td>Default break text (*)</td>
<td>Select this check box to generate break footing text in breaks for which you did not indicate break footing text.</td>
</tr>
<tr>
<td></td>
<td>The default break text consists of one asterisk for the highest numbered break level text, two asterisks for the next-highest numbered break level text, and so on.</td>
</tr>
<tr>
<td>Function name in column heading when grouping</td>
<td>Select this check box to add the name of the summarization function to the heading of the aggregated columns, if a report has summarized columns and you use the GROUP usage code to suppress the tabular data lines.</td>
</tr>
<tr>
<td>Column wrapped lines kept on a page</td>
<td>This field applies only to forms created using the ISPF version of the application QMF for TSO/CICS. Select this check box to split wrapped columns between two pages, if you specified column wrapping for one or more columns.</td>
</tr>
<tr>
<td>Across summary column</td>
<td>Select this check box to display the automatically generated ACROSS summary column, which produces additional columns that summarize (total) across the specified columns.</td>
</tr>
</tbody>
</table>
### Table 24. Options section elements for a classic report (continued)

<table>
<thead>
<tr>
<th>Element</th>
<th>Description</th>
</tr>
</thead>
</table>
| **Automatic reordering of report columns** | Select this check box to automatically reorder the columns in a report when you specify a BREAK or GROUP usage code, or one of the aggregating usage codes.  
If selected, the columns are reordered so that BREAK columns are to the far left, GROUP columns are to the left after BREAK columns, all non-aggregated columns are to the left after BREAK and GROUP columns, and all aggregated columns are to the far right. |
| **Page renumbering at the highest break level** | Select this check box if a printed report should begin a new page beginning with the number 1 whenever the value in the control column of the highest break level changes.  
**Note:** The highest break level is the one with the lowest number. |
| **Column heading separator**          | Select this check box to display a row of dashed lines between the column headings and the tabular data lines.                                                                                               |
| **Across heading separator**          | Select this check box to mark columns in ACROSS reports with dashed lines and arrows.                                                                                                                      |
| **Break summary separator**           | Select this check box to display a row of equal signs between the tabular data lines and the break summary.                                                                                                |
| **Final summary separator**           | Select this check box to display a row of equal signs between the tabular data lines and the final summary.                                                                                                 |
| **Place LOB contents inline**         | Select this check box to display LOB contents inline in the contents of the form.                                                                                                                           |

### Specifying Final formatting options

You use the **Final** formatting options to define the content and placement of your report's final text. For example, you can include final text and summary data at the end of the report.

### About this task

#### Table 25. Final section elements for a classic report

<table>
<thead>
<tr>
<th>Element</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>New page for final text</strong></td>
<td>Check this check box to start a new report page for the final text.</td>
</tr>
<tr>
<td><strong>Put final summary at line</strong></td>
<td>Specify where the final summary line is placed vertically in the final text lines. The range is between 1 and 999. If you do not want a final summary line, specify NONE.</td>
</tr>
<tr>
<td><strong>Blank lines before text</strong></td>
<td>Specify the number of blank lines before the first and final text line. The range is between 0 and 999 or the word BOTTOM, which is treated as a zero.</td>
</tr>
</tbody>
</table>
Table 25. Final section elements for a classic report (continued)

<table>
<thead>
<tr>
<th>Element</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Text</td>
<td>You use the buttons above the Final text table to add, edit, or remove lines, and also change the lines order. Specify the lines of final text:</td>
</tr>
<tr>
<td></td>
<td>• <strong>Line</strong>: specify where the final text is placed vertically. You do not have to start with line 1 and you do not have to specify consecutive line numbers.</td>
</tr>
<tr>
<td></td>
<td>• <strong>Alignment</strong>: specify where the text is placed horizontally on the line. Valid alignment values are LEFT, RIGHT, CENTER, APPEND, or the number of a specific position in the line. If you specify APPEND, the text is placed immediately following the previous line of text.</td>
</tr>
<tr>
<td></td>
<td>• <strong>Text</strong>: type the text that appears at the bottom of the report. You can use form variables, global variables and HTML variables in this field.</td>
</tr>
</tbody>
</table>

Specifying Page formatting options
You use the Page options to define the content and placement of the page heading and page footing on your report.

About this task

Table 26. Page section elements for a classic report

<table>
<thead>
<tr>
<th>Element</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blank lines before heading</td>
<td>Specify the number of blank lines before the first page heading line. The range is between 0 and 999.</td>
</tr>
<tr>
<td>Blank lines after heading</td>
<td>Specify the number of blank lines after the last page heading line. The range is between 0 and 999.</td>
</tr>
<tr>
<td>Page heading text</td>
<td>You use the buttons above the Page heading text table to add, edit, or remove lines, and also change the lines order. Type each line of page heading text.</td>
</tr>
<tr>
<td></td>
<td>• <strong>Line</strong>: specify where to place the final text vertically. You do not have to start with line 1 and you do not have to specify consecutive line numbers.</td>
</tr>
<tr>
<td></td>
<td>• <strong>Alignment</strong>: specify where to place the text horizontally on the line. Valid alignment values are LEFT, RIGHT, CENTER, APPEND, or the number of a specific position in the line. If you specify APPEND, the text is placed immediately following the previous line of text.</td>
</tr>
<tr>
<td></td>
<td>• <strong>Text</strong>: type the final text that appears in the page heading on the report. You can use form variables, global variables and HTML variables in this field.</td>
</tr>
<tr>
<td>Blank lines before footing</td>
<td>Specify the number of blank lines before the first page footing line. The range is between 0 and 999.</td>
</tr>
<tr>
<td>Blank lines after footing</td>
<td>Specify the number of blank lines after the last page footing line. The range is between 0 and 999.</td>
</tr>
</tbody>
</table>
## Table 26. Page section elements for a classic report (continued)

<table>
<thead>
<tr>
<th>Element</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Page footing text</td>
<td>You use the buttons above the Page footing text table to add, edit, or remove lines, and also change the lines order. Specify each line of text printed in the page footing at the bottom of the page:</td>
</tr>
<tr>
<td></td>
<td>• <strong>Line</strong>: specify where to place the final text vertically. You do not have to start with line 1 and you do not have to specify consecutive line numbers.</td>
</tr>
<tr>
<td></td>
<td>• <strong>Alignment</strong>: specify where to place the text horizontally on the line. Valid alignment values are LEFT, RIGHT, CENTER, APPEND, or the number of a specific position in the line. If you specify APPEND, the text is placed immediately following the previous line of text.</td>
</tr>
<tr>
<td></td>
<td>• <strong>Text</strong>: type the page footing text that appears at the bottom of the report.</td>
</tr>
</tbody>
</table>

### Using form variables

Form variables are codes that you can insert into text fields to produce information on the report.

For example, you can insert a date variable to produce the current date whenever the report is printed. The following form variables are available:

- **&ROW**: This variable will display the number of the current row of data.
- **&DATE**: This variable will display the current date.
- **&TIME**: This variable will display the current time.
- **&PAGE**: This variable will display the current page number.
- **&COUNT**: This variable will display the number of rows retrieved or printed since the last break at the same level.
- **&CALCid**: This variable identifies a Form calculation expression to use, where "id" is the ID number of the expression.
- **&n**: This variable will display the value of a column, where "n" is the column number.
- **&an**: This variable will display the aggregation of a column, where "n" is the column number and "a" is one of the following aggregation variables: AVG, COUNT, CPCT, CSUM, FIRST, LAST, MAX, MIN, PCT, STDEV, SUM, TCPCT, or TPCT. The aggregation is based on the rows retrieved or printed since the last break at the same level.
- **&Global Variables**: This will display the value of the global variable.
- **&HTML Variables**: This will display the value of the HTML variable.

Different variables are available for use depending on the part of the form that you are editing. The following list shows the types of variables that you can use in each part of a form:

- **Page Heading**: You can use all variables except &an, &COUNT, or &CALCid.
- **Page Footing**: You can use all variables except &an, &COUNT, or &CALCid.
- **Break Heading**: You can use all variables except &an, &COUNT, or &CALCid.
- **Break Footing**: You can use all variables.
- **Calculation Expression**: You can use all variables except &CALCid.
- **Column Expression**: You can use all variables except &Page, &an, &COUNT, or &CALCid.
- **Condition**: You can use all variables except &Page, &an, &COUNT, or &CALCid.
- **Detail Heading**: You can use all variables except &COUNT or &CALCid.
- **Detail Block**: You can use all variables.
- **Final Text**: You can use all variables.

**Using usage codes**

Usage codes provide summary information about the data in a column.

For example, usage codes can provide total summary information at the end of a column, or partial summaries at control breaks in a table. The usage codes available depend on the data in the column and the type of summary.

<table>
<thead>
<tr>
<th>Usage Code</th>
<th>Description</th>
<th>Notes</th>
</tr>
</thead>
</table>
| ACROSS     | Enables you to produce a report with horizontal control breaks. | In an ACROSS report:  
- The number and titles of the columns in the report are dependent on the values in the ACROSS column. There is one set of report columns for each value in the ACROSS column and the header for each is the value of column. The set of report columns includes a column for each one that uses an aggregation usage code (for example, SUM, AVERAGE, COUNT).  
- You can only have one ACROSS column in a report.  
- The number of rows and the title of each row in the report are dependent on the values in the GROUP column(s). There is one row for each value in the GROUP column(s) and the title of each row is the value of the column(s).  
- The CSUM, PCT, CPCT, TPCT and TCPCT usage codes are only partially supported when generating reports that also use the ACROSS usage code. |
<p>| AVERAGE    | Average of the values in the column | This usage code is only valid for numeric data. This calculated value appears as a total in the report. The calculated value is formatted with the edit code of the column. |
| Breakn     | Provide a control break level (where &quot;n&quot; represents a number between 1 and 6). | For example, BREAK1 specifies a control column for a level-1 break and BREAK2 specifies a control column for a level-2 break. Any change in the value of the column causes a section break in the report. Subtotals are displayed for columns whose usage is one of the aggregation usages. Also, the text specified the appropriate Form Break component is displayed. Your query should use an ORDER BY clause that matches your BREAK columns. |</p>
<table>
<thead>
<tr>
<th>Usage Code</th>
<th>Description</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>BREAKnX</td>
<td>Same as BREAKn, except the control column is omitted from the report.</td>
<td>Same as BREAKn.</td>
</tr>
<tr>
<td>CALCid</td>
<td>The evaluation of the calculation expression in the Form calculations component where ID equals &quot;id&quot;.</td>
<td>This calculated value appears as a total in the report and applies only to the last row of data. The calculated value is formatted with the edit code of the column. If the column value is used in the calculation, only the last row of data is evaluated.</td>
</tr>
<tr>
<td>COUNT</td>
<td>Count of the non-null values in the column.</td>
<td>This calculated value appears as a total in the report. The calculated value is formatted with the edit code K.</td>
</tr>
</tbody>
</table>
| CPCT       | Cumulative percentage each value of the column is of the current total. | • This calculated value replaces each detail line value and also appears as a total in the report.  
• The calculated value is formatted with the edit code of the column.  
• The CPCT usage code is only partially supported when generating reports that also use the ACROSS usage code. |
| CSUM       | Cumulative sum of the values in the column. | • This calculated value replaces each detail line value and also appears as a total in the report.  
• The calculated value is formatted with the edit code of the column.  
• The CSUM usage code is only partially supported when generating reports that also use the ACROSS usage code. |
| FIRST      | First value of the column. | This calculated value appears as a total in the report. The calculated value is formatted with the edit code of the column. |
| GROUP      | Displays only one line of summary data for each set of values in the column. | More than one column can have usage code GROUP. If so, a change in value in any column starts a new group. All other columns with no usage code are omitted from the report. |
| LAST       | Last value in the column. | This calculated value appears as a total in the report. The calculated value is formatted with the edit code of the column. |
| MAXIMUM    | Maximum value in the column. | This calculated value appears as a total in the report. The calculated value is formatted with the edit code of the column. |
| MINIMUM    | Minimum value in the column. | This calculated value appears as a total in the report. The calculated value is formatted with the edit code of the column. |
| OMIT       | Excludes the column from the report. | The column and its values are not included in the tabular report. The values in the column can still appear in the report (for example, in a break footer) by use of form variables (such as &n, where n represents the column number). |
### Saving classic reports

When you save a classic report you are actually saving the form that is used to create the report. The form is saved as an object and can be saved in a local or network file, in your workspace, in a QMF catalog, or in a repository.

### About this task

Saved forms open differently depending on whether the form was saved with an associated data source object.

- When you save the form with an associated data source object then you are saving both elements that are used to create the report. Upon opening a form that has been saved with both the data source object and the form: the data source object is located; the data source object is run and the query results are obtained; the form formatting is applied; and the classic report is presented. Specifying that a particular data source object will always be used when a form is opened streamlines the steps required to create a classic report.

- When you save the form without an associated data source object, then you are only saving the formatting options. When you open the form, you will must...
supply the query results. That means you must run a query, make the query results active and open the form in order to create the classic report. If no query results are available then the form opens only for editing.

**Saving a form with an associated data source object**

When you save the form with an associated data source object then both elements used to create the report are saved. This method of saving streamlines the steps required to create a classic report.

**About this task**

To save a form with an associated data source object:

**Procedure**

1. Create a query object and save the query object or open a saved query object.
2. Run the query object to obtain query results.
3. Generate a classic report using a new or saved form; or default form formatting or formatting based on the query results.
4. Save the form. When you open this saved form, it has a known associated data source object (because you saved the object). The data source object is located, the data source object is run and the query results are obtained, the form formatting is applied and the classic report is presented.

**Saving a form without an associated data source object**

When you save the form without an associated data source object, then you only save the formatting options.

**About this task**

To save a form without an associated data source object:

1. Create a query object or open an existing query.
2. Run the query object to obtain query results.
3. Generate a classic report using either default settings, formatting settings based on the query results, opening a saved form, or creating a new form.
4. Select Form > Data Source. Clear the Always use this object when opening this document check box.
5. Save the form. When you open this saved form, it has no known associated data source object. If a set of query results are active, they will be formatted into a classic report, otherwise the form will open for editing.

**Working with quick reports**

You use quick reports to rapidly generate a printable report from a query result set.

You use the Create New Quick Report wizard to define the report ‘break’ sections, determine the columns over which data will be reported. For example, report sales by region, followed by country within each region. In the Create New Quick Report wizard, you can also define the aggregation function applied to each column value (ascending, descending, count, maximum, minimum, sum, average, and so on), as well to define the formatting (text, alignment, font, background color, and so on) of both the detail and summary rows of the report.
Creating quick reports

You can create a quick report from the data in a query by using the Create New Quick Report wizard.

About this task

To create a quick report, perform the following procedure:

Procedure

1. To open the Create New Quick Report wizard, select File > New > Other, expand the DataQuant Objects and select Quick Report.
2. On the first page of the Create New Quick Report wizard, specify the quick report name and select the required source query. Select the Copy settings from grid check box to apply such grid settings as grouping and detail columns distribution and formatting options to a quick report. Click Next.

Note: You can create a quick report on the basis of an opened query result set. With an active query in the editor window, select Query > Transfer To > Quick Report. In this case, the first page of the Create New Quick Report wizard is skipped and the Columns page is opened automatically.
4. On the Format page of the wizard, set the following quick report options:
   - Report title and total line format options
   - Columns order
   - Column headers format options
   - Column groups
   - Total and detail columns sets
   - Conditional format for report columns
   - Grouping columns format options
   - Header and Footer of grouping columns
   - Detail columns format options
   - Custom data objects that can be assigned to report columns.
5. Click Finish. The quick report is displayed on the Design page of the editor where you can manually customize the look of the quick report. You can view the print version of the quick report on the Preview page of the editor.

Arranging grouping and detail columns

You can add and remove the grouping and detail columns from a quick report and convert a grouping column to a detail column and vice versa.

Before you begin

You must open the Create New Quick Report wizard.

About this task

To arrange the grouping and detail columns in a quick report:

Procedure

1. Open the Columns page of the Create New Quick Report wizard.
2. To add or remove grouping columns, click **Move to grouping columns set** or **Remove from grouping columns set**.

3. To add or remove detail columns, click **Move to detail columns set** or **Remove from detail columns set**.

   **Note:** You can add all available columns to the **Detail Columns** list by clicking **Move all to detail columns set**. To remove all columns from the **Detail Columns** list, click **Move all from detail columns set**.

4. To convert a grouping column to detail column and vice versa, select the columns and then click **Move to grouping columns set** or **Move to detail columns set** buttons.

5. To change the order of grouping columns, use **Move selected columns up** or **Move selected columns down** buttons to the right of the **Grouping Columns** list.

6. If you have specified all the required parameters for the quick report, click **Finish** to close the Create New Quick Report wizard and display the quick report on the **Design** page of the editor where you can manually customize the look of the report.

**What to do next**

**Next:** “Specifying how to display the report title and report total information” on page 182

**Related tasks:**

“Specifying how to display the report title and report total information” on page 182

You can specify how to display the report title and report total information of a quick report.

“Specifying how to display column headers” on page 183

You can specify the look of column headers and the order in which they are displayed in a quick report on the **Columns Headers** page of the Create New Quick Report wizard.

“Creating column groups” on page 184

To limit the data that is displayed in a quick report, you can organize the columns in mutually exclusive groups of columns.

“Creating total and detail column sets” on page 186

You can associate detail columns that you do not always want to be displayed in a quick report with a total column.

“Applying conditional formatting” on page 187

You can use a conditional format in quick reports to help you visually explore and analyze data, discover critical issues, and identify patterns and trends.

“Specifying format options for the Grouping Columns” on page 190

You can specify format options for the Grouping Columns of a quick report.

“Specifying how to display Header and Footer information” on page 191

You can specify the how to display Header and Footer information for a grouping column of a quick report.

“Specifying format options for the Details Columns” on page 192

You can specify how to display the Details Columns of a quick report.

“Specifying quick report column format options” on page 194

You can specify the formatting options of individual columns of your quick report.
You can assign custom data objects that you want to be opened when you click a column in your quick report.

**Specifying how to display the report title and report total information**
You can specify how to display the report title and report total information of a quick report.

**Before you begin**
You must open the Create New Quick Report wizard.

**About this task**
To specify how to display the report title and report total information in the quick report:

**Procedure**
1. Open the Format page of the Create New Quick Report wizard.
2. Select Report Settings from the tree.
3. Select the Visible check box in the Report Title section to display the title of the report. The Report Title options become available.
4. Specify the title of the report in the Text field.
   
   **Note:** You can view the quick report title only on the printed report.
5. From the Alignment menu, specify where on the page the report title must be displayed. Valid options are:
   - **Left** - The title is displayed on the left hand side of the header.
   - **Center** - The title is displayed in the center of the header.
   - **Right** - The title is displayed on the right hand side of the header.
6. Specify the font of the title in the Font field.
7. Specify the background color of the title in the Background field.
8. Select the Visible check box in the Total Line section to include a totals line at the bottom of the report. The Total Line options become available.
9. Specify the font of the totals line text in the Font field.
10. Specify the background color of the totals line in the Background field.
11. In the Common settings area, specify how to display the quick report grid:
    - To display row and column numbers, select the Show row and column numbers check box.
    - To display grid lines, select the Show grid lines check box.
    
    By default, both check boxes are cleared.
12. If you have specified all the required parameters for the quick report, click Finish to close the Create New Quick Report wizard and display the quick report on the Design page of the editor where you can manually customize the look of the report.

**What to do next**

Next: “Specifying how to display column headers” on page 183

**Related tasks:**
You can add and remove the grouping and detail columns from a quick report and convert a grouping column to a detail column and vice versa.

You can specify the look of column headers and the order in which they are displayed in a quick report on the Columns Headers page of the Create New Quick Report wizard.

To limit the data that is displayed in a quick report, you can organize the columns in mutually exclusive groups of columns.

You can associate detail columns that you do not always want to be displayed in a quick report with a total column.

You can use a conditional format in quick reports to help you visually explore and analyze data, discover critical issues, and identify patterns and trends.

You can specify format options for the Grouping Columns of a quick report.

You can specify how to display Header and Footer information for a grouping column of a quick report.

You can specify how to display the Details Columns of a quick report.

You can specify the formatting options of individual columns of your quick report.

You can assign custom data objects that you want to be opened when you click a column in your quick report.

Specifying how to display column headers

You can specify the look of column headers and the order in which they are displayed in a quick report on the Columns Headers page of the Create New Quick Report wizard.

**Before you begin**

You must open the Create New Quick Report wizard.

**About this task**

To specify how to display column headers:

**Procedure**

1. Open the Format page of the Create New Quick Report wizard.
2. Expand Report Settings node and select Column Headers node from the tree.
3. To change the order of the columns, select a column and drag it to the required position.
4. To hide a column, select this column and then select the Hide check box.
5. To create hierarchical column headers or group existing column headers, click Create Header, and drag the required column headers to this header.
Note: If you create a header and do not populate it with at least one of existing headers, it will not be displayed in the report.

6. In the **Header** area, specify the name, font, background color, horizontal and vertical alignment of the column header.

7. If you have specified all the required parameters for the quick report, click **Finish** to close the Create New Quick Report wizard and display the quick report on the **Design** page of the editor where you can manually customize the look of the report.

**What to do next**

Next: “Creating column groups”

**Related tasks:**

- “Arranging grouping and detail columns” on page 180
  You can add and remove the grouping and detail columns from a quick report and convert a grouping column to a detail column and vice versa.

- “Specifying how to display the report title and report total information” on page 182
  You can specify how to display the report title and report total information of a quick report.

- “Creating column groups”
  To limit the data that is displayed in a quick report, you can organize the columns in mutually exclusive groups of columns.

- “Creating total and detail column sets” on page 186
  You can associate detail columns that you do not always want to be displayed in a quick report with a total column.

- “Applying conditional formatting” on page 187
  You can use a conditional format in quick reports to help you visually explore and analyze data, discover critical issues, and identify patterns and trends.

- “Specifying format options for the Grouping Columns” on page 190
  You can specify format options for the Grouping Columns of a quick report.

- “Specifying how to display Header and Footer information” on page 191
  You can specify the how to display Header and Footer information for a grouping column of a quick report.

- “Specifying format options for the Details Columns” on page 192
  You can specify how to display the Details Columns of a quick report.

- “Specifying quick report column format options” on page 194
  You can specify the formatting options of individual columns of your quick report.

- “Assigning custom data objects to quick report columns” on page 196
  You can assign custom data objects that you want to be opened when you click a column in your quick report.

**Creating column groups**

To limit the data that is displayed in a quick report, you can organize the columns in mutually exclusive groups of columns.

**Before you begin**

You must open the Create New Quick Report wizard.
About this task

You can then toggle between the groups to display only those columns that are assigned to that group. Columns assigned to other groups are not displayed. Any columns not assigned to any group are always displayed in the quick report.

To create a column group:

Procedure

1. Open the Format page of the Create New Quick Report wizard.
2. Expand Report Settings node and select Column Groups node from the tree.
3. On the Column Groups page, click Add group. A new group is displayed in the Groups list.
4. Add the required columns to the group by selecting them from the Columns list.
5. Optional: To hide the columns already assigned to a group from the Columns list, clear the Show grouped columns check box.
6. Optional: To change the order of the column groups, select a group and click Move Up or Move Down.

Note: The first group in the list is displayed by default when you open the quick report.
7. If you have specified all the required parameters for the quick report, click Finish to close the Create New Quick Report wizard and display the quick report on the Design page of the editor where you can manually customize the look of the report.

What to do next

Next: “Creating total and detail column sets” on page 186

Related tasks:

“Arranging grouping and detail columns” on page 180
You can add and remove the grouping and detail columns from a quick report and convert a grouping column to a detail column and vice versa.

“Specifying how to display the report title and report total information” on page 182
You can specify how to display the report title and report total information of a quick report.

“Specifying how to display column headers” on page 183
You can specify the look of column headers and the order in which they are displayed in a quick report on the Columns Headers page of the Create New Quick Report wizard.

“Creating total and detail column sets” on page 186
You can associate detail columns that you do not always want to be displayed in a quick report with a total column.

“Applying conditional formatting” on page 187
You can use a conditional format in quick reports to help you visually explore and analyze data, discover critical issues, and identify patterns and trends.

“Specifying format options for the Grouping Columns” on page 190
You can specify format options for the Grouping Columns of a quick report.

“Specifying how to display Header and Footer information” on page 191
You can specify the how to display Header and Footer information for a grouping
column of a quick report.

“Specifying format options for the Details Columns” on page 192
You can specify how to display the Details Columns of a quick report.

“Specifying quick report column format options” on page 194
You can specify the formatting options of individual columns of your quick report.

“Assigning custom data objects to quick report columns” on page 196
You can assign custom data objects that you want to be opened when you click a column in your quick report.

Creating total and detail column sets
You can associate detail columns that you do not always want to be displayed in a quick report with a total column.

Before you begin
You must open the Create New Quick Report wizard.

About this task
For example, you might have a Product Costs column that is calculated from the Item Cost and Number of Items Shipped. In the quick report you might want only the Product Costs column to be displayed, but you still want the other columns available if needed. You can use this feature to make the Product Costs column the total column, and specify the columns used to calculate the total column as detail columns.

You can display or hide detail columns by clicking the arrow in the total column header.

To create total and detail column set:

Procedure
1. Open the Format page of the Create New Quick Report wizard.
2. Expand Report Settings node and select Total/Detail Sets node from the tree.
3. On the Total/Detail Sets page, click Add Total/Detail Set. A new set is displayed in the table.
4. Specify a total column from the list of available columns. All of the non-selected columns are automatically displayed in the Detail Columns list.
5. From the Detail Columns list, select detail columns to associate them with the specified total column.
6. If you have specified all the required parameters for the quick report, click Finish to close the Create New Quick Report wizard and display the quick report on the Design page of the editor where you can manually customize the look of the report.

What to do next
Next: “Applying conditional formatting” on page 187
Related tasks:
“Arranging grouping and detail columns” on page 180
You can add and remove the grouping and detail columns from a quick report and convert a grouping column to a detail column and vice versa.
Applying conditional formatting

You can use a conditional format in quick reports to help you visually explore and analyze data, discover critical issues, and identify patterns and trends.

**Before you begin**

You must open the Create New Quick Report wizard.

**About this task**

Conditional format allows you to specify conditional expressions that control both the display of the data and the behavior of a quick report based on the underlying data. For example, you can use conditional format to highlight in red any fields that indicate year-end sales that fall below a particular figure.

To apply conditional formatting:

**Procedure**

1. Open the Format page of the Create New Quick Report wizard.
2. Expand **Report Settings** node and select **Conditional Format** node from the tree.
3. On the **Conditional Format** page, click **Add Format**.
4. On the first page of the New Conditional Format wizard, specify the conditional format name and select the columns and levels of detail to which this format will be applied. Click **Next**.
**Note:** If the format for the selected columns or levels of detail has already been set, the new format will override it.

5. On the second page of the wizard, specify conditional expressions and the format options.

6. From the **Base column** list, select the column which values will be used as a part of condition for formatting. You can select the <**Based on the same column**> option, to use the values from the column to which you apply the conditional format. If you select this option, the format can only be applied to the columns of the same type (numeric, string, or date/time).

7. Click **Add Condition** and specify condition parameters in the **Selected condition** area.
   a. From the **Operator** list, select the required operator. Available options are:
      - **Not Less Than** - All values that are equal to or greater than the specified value are included.
      - **Greater Than** - All values that are greater than the specified value are included.
      - **Not Greater Than** - All values that are equal to or less than the specified value are included.
      - **Less Than** - All values that are less than the specified value are included.
      - **Equal To** - Only values that are equal to the specified value are included.
      - **Not Equal To** - All values that are not equal to the specified value are included.
      - **Like** - All values that match the specified pattern are included. The pattern can contain the following special characters:
        - % matches zero (0) or more characters.
        - _ matches only one (1) character.
        - ? matches only one (1) character.
      - **Unlike** - All values that are not like the specified value are included.
      - **Is Null** - Only null values are included.
      - **Is Not Null** - All of the values that are not null are included.
      - **In Range** - All of the values that fall within the two specified values, along with the values themselves, are included.
      - **Outside Range** - All of the values that fall outside of the two values, along with the values themselves, are included.
      - **Between** - All of the values that fall within the two specified values are included.
      - **Not Between** - All of the values that fall outside of the two specified values are included.
   b. Specify the types and values for the operands. You can set either a constant value or a name of the column that will provide required values.

8. In the editor, specify the conditional format options. You can preview the resulting format in the **Sample** area.
   a. On the **Text** page of the editor, set the text font, data format, and alignment.
   b. On the **Colors** page, set the foreground and background colors. You can use either a single color or gradient.
c. On the **Image** page, you can select an image to display in the report and set its position. Available position options are:
   - Before text
   - After text
   - Replace text.

9. Click **Finish** to close the New Conditional Format wizard. The created condition format is displayed on the **Conditional Format** page of the Create New Quick Report wizard.

10. If you have specified all the required parameters for the quick report, click **Finish** to close the Create New Quick Report wizard and display the quick report on the **Design** page of the editor where you can manually customize the look of the report.

**Results**

You can edit created conditional formats by using the Edit Conditional Format wizard. You can also reassign created conditional formats to other columns or levels of detail on the **Assignments** page.

**What to do next**

Next: “Specifying format options for the Grouping Columns” on page 190

Related tasks:

- “Arranging grouping and detail columns” on page 180
  You can add and remove the grouping and detail columns from a quick report and convert a grouping column to a detail column and vice versa.

- “Specifying how to display the report title and report total information” on page 182
  You can specify how to display the report title and report total information of a quick report.

- “Specifying how to display column headers” on page 183
  You can specify the look of column headers and the order in which they are displayed in a quick report on the **Columns Headers** page of the Create New Quick Report wizard.

- “Creating total and detail column sets” on page 186
  You can associate detail columns that you do not always want to be displayed in a quick report with a total column.

- “Specifying format options for the Grouping Columns” on page 190
  You can specify format options for the Grouping Columns of a quick report.

- “Specifying how to display Header and Footer information” on page 191
  You can specify the how to display Header and Footer information for a grouping column of a quick report.

- “Specifying format options for the Details Columns” on page 192
  You can specify how to display the Details Columns of a quick report.

- “Specifying quick report column format options” on page 194
  You can specify the formatting options of individual columns of your quick report.

- “Assigning custom data objects to quick report columns” on page 196
  You can assign custom data objects that you want to be opened when you click a column in your quick report.
Specifying format options for the Grouping Columns
You can specify format options for the Grouping Columns of a quick report.

Before you begin
You must open the Create New Quick Report wizard.

About this task
To specify Grouping Columns formatting options:

Procedure
1. Open the Format page of the Create New Quick Report wizard.
2. Select Grouping Columns from the tree.
3. To change a grouping column’s name, click the Header cell and specify the new text.
   The text that you specify will display as the grouping column name in the quick report. To set a multiline header you should use “_” symbol to separate lines. In the report it will be substituted by the line break character.
4. To change the sorting order of a grouping column, click the Sort cell and select Ascending or Descending.
5. To change the width of a grouping column, click the Width cell and specify the new width.
6. If you have specified all the required parameters for the quick report, click Finish to close the Create New Quick Report wizard and display the quick report on the Design page of the editor where you can manually customize the look of the report.

What to do next
Next: “Specifying how to display Header and Footer information” on page 191
Related tasks:
“Arranging grouping and detail columns” on page 180
You can add and remove the grouping and detail columns from a quick report and convert a grouping column to a detail column and vice versa.
“Specifying how to display the report title and report total information” on page 182
You can specify how to display the report title and report total information of a quick report.
“Specifying how to display column headers” on page 183
You can specify the look of column headers and the order in which they are displayed in a quick report on the Columns Headers page of the Create New Quick Report wizard.
“Applying conditional formatting” on page 187
You can use a conditional format in quick reports to help you visually explore and analyze data, discover critical issues, and identify patterns and trends.
“Specifying how to display Header and Footer information” on page 191
You can specify the how to display Header and Footer information for a grouping column of a quick report.
“Specifying format options for the Details Columns” on page 192
You can specify how to display the Details Columns of a quick report.
“Specifying quick report column format options” on page 194
You can specify the formatting options of individual columns of your quick report.

“Assigning custom data objects to quick report columns” on page 196
You can assign custom data objects that you want to be opened when you click a column in your quick report.

**Specifying how to display Header and Footer information**
You can specify the how to display Header and Footer information for a grouping column of a quick report.

**Before you begin**
You must open the Create New Quick Report wizard.

**About this task**
To specify Header and Footer information for the **Grouping Columns** options:

**Procedure**
1. Open the Format page of the Create New Quick Report wizard.
2. In the tree, expand **Grouping Columns**, then one of the column names under it, and select **Header**.
3. To display a header for every grouping column row, select the **Visible** check box.
4. Specify the text that you want to display in the grouping column row header in the **Text** field. The default text is ="column_name: '@[column]" where 'column_name' is the column name of the grouping column and 'column' is the value of the given grouping column row. For example, if your grouping column is named DEPT and it contains department numbers of a given company, this text could generate sequential row headers of DEPT: 10, DEPT: 20, and so on. You can change this text to anything that you want, but it is recommended that you keep this overall structure.
5. Specify the font of the grouping column row headers in the **Font** field.
6. Specify the background color of the grouping column row headers in the **Background** field.
7. Specify the height of the header in the **Height** field.
8. You can display charts in the Header of a grouping column. To display a chart in the header, select **Enable chart** check box. In the New chart wizard specify parameters of the chart.

**Note:** If you clear the **Enable chart** check box, the specified chart will not be displayed but it is not removed from the quick report.
9. You can edit a created chart in the Edit chart properties window. To open the Edit chart properties window, click **Chart settings**.
10. To remove a chart, click **Delete chart**.
11. The **Footer** options are identical to the **Header** options. To open the Footer page, select **Footer** in the tree.
12. If you want to display the aggregate totals of the specified detail columns, select the **Visible** check box and specify the totals line text in the **Text** field. The default text is ="Summary of ' @[column]" where 'column' is the name of a details column which values should be aggregated.
13. If you have specified all the required parameters for the quick report, click **Finish** to close the Create New Quick Report wizard and display the quick report on the **Design** page of the editor where you can manually customize the look of the report.

**What to do next**

**Next:** “Specifying format options for the Details Columns”

**Related tasks:**
- “Creating charts” on page 217
  You can create a chart from a query result set with the help of the New Chart wizard.
- “Arranging grouping and detail columns” on page 180
  You can add and remove the grouping and detail columns from a quick report and convert a grouping column to a detail column and vice versa.
- “Specifying how to display the report title and report total information” on page 182
  You can specify how to display the report title and report total information of a quick report.
- “Specifying how to display column headers” on page 183
  You can specify the look of column headers and the order in which they are displayed in a quick report on the **Columns Headers** page of the Create New Quick Report wizard.
- “Creating column groups” on page 184
  To limit the data that is displayed in a quick report, you can organize the columns in mutually exclusive groups of columns.
- “Creating total and detail column sets” on page 186
  You can associate detail columns that you do not always want to be displayed in a quick report with a total column.
- “Applying conditional formatting” on page 187
  You can use a conditional format in quick reports to help you visually explore and analyze data, discover critical issues, and identify patterns and trends.
- “Specifying format options for the Grouping Columns” on page 190
  You can specify format options for the Grouping Columns of a quick report.
- “Specifying format options for the Details Columns”
  You can specify how to display the Details Columns of a quick report.
- “Specifying quick report column format options” on page 194
  You can specify the formatting options of individual columns of your quick report.
- “Assigning custom data objects to quick report columns” on page 196
  You can assign custom data objects that you want to be opened when you click a column in your quick report.

**Specifying format options for the Details Columns**

You can specify how to display the Details Columns of a quick report.

**Before you begin**

You must open the Create New Quick Report wizard.

**About this task**

To specify **Details Columns** formatting options:
Procedure
1. Open the Format page of the Create New Quick Report wizard.
2. Select Details Columns from the tree. The Details Columns page opens.
3. To change a detail column's column name, click the Header cell and specify the new text. The text that you specify will display as the detail column name in the quick report. To set a multiline header you should use “_” symbol to separate lines. In the report it will be substituted by the line break character.
4. To change how a detail column is aggregated, click the Aggregate cell and select a new value. Valid options include:
   - Count
   - First
   - Last
   - Maximum
   - Minimum
   - Sum
   - Average
   - Standard Deviation
   Select None to remove aggregation from a column.
5. To change the width of a detail column, click the Width cell and specify the new width.
6. If you have specified all the required parameters for the quick report, click Finish to close the Create New Quick Report wizard and display the quick report on the Design page of the editor where you can manually customize the look of the report.

What to do next

Next: “Specifying quick report column format options” on page 194

Related tasks:
“Arranging grouping and detail columns” on page 180
You can add and remove the grouping and detail columns from a quick report and convert a grouping column to a detail column and vice versa.

“Specifying how to display the report title and report total information” on page 182
You can specify how to display the report title and report total information of a quick report.

“Specifying how to display column headers” on page 183
You can specify the look of column headers and the order in which they are displayed in a quick report on the Columns Headers page of the Create New Quick Report wizard.

“Creating column groups” on page 184
To limit the data that is displayed in a quick report, you can organize the columns in mutually exclusive groups of columns.

“Creating total and detail column sets” on page 186
You can associate detail columns that you do not always want to be displayed in a quick report with a total column.

“Applying conditional formatting” on page 187
You can use a conditional format in quick reports to help you visually explore and analyze data, discover critical issues, and identify patterns and trends.
You can specify format options for the Grouping Columns of a quick report.

You can specify how to display Header and Footer information for a grouping column of a quick report.

You can specify the formatting options of individual columns of your quick report.

You can assign custom data objects that you want to be opened when you click a column in your quick report.

**Specifying quick report column format options**
You can specify the formatting options of individual columns of your quick report.

**About this task**

In addition to customizing the look and feel of a quick report's summary rows, you can specify the formatting options for each column that is displayed in your report.

To specify individual column formatting options:

**Procedure**

1. Open the Format page of the Create New Quick Report wizard.
2. Select a column name from the tree. The formatting options for each row are identical, whether the column is a grouping column or a detail column.
3. Specify the width of the column in the **Column Width** field.
4. In the Details area, specify the font, data format, background color, horizontal and vertical alignment of the column. For the columns containing character data, you can apply automatic text wrapping. To do this, select the **Wrap text** check box.
5. If you have specified all the required parameters for the quick report, click **Finish** to close the Create New Quick Report wizard and display the quick report in the **Preview** window.

**Related tasks:**

“Arranging grouping and detail columns” on page 180
You can add and remove the grouping and detail columns from a quick report and convert a grouping column to a detail column and vice versa.

“Specifying how to display the report title and report total information” on page 182
You can specify how to display the report title and report total information of a quick report.

“Specifying how to display column headers” on page 183
You can specify the look of column headers and the order in which they are displayed in a quick report on the **Columns Headers** page of the Create New Quick Report wizard.

“Creating column groups” on page 184
To limit the data that is displayed in a quick report, you can organize the columns in mutually exclusive groups of columns.

“Creating total and detail column sets” on page 186
You can associate detail columns that you do not always want to be displayed in a quick report with a total column.
You can use a conditional format in quick reports to help you visually explore and analyze data, discover critical issues, and identify patterns and trends.

You can specify format options for the Grouping Columns of a quick report.

You can specify how to display Header and Footer information for a grouping column of a quick report.

You can specify how to display the Details Columns of a quick report.

You can assign custom data objects that you want to be opened when you click a column in your quick report.

Specifying data format options for quick report columns:

You can specify data values format options for individual columns in the Choose a format window.

About this task

To specify data format options:

Procedure
1. Open the Format page of the Create New Quick Report wizard.
2. Select a column name from the tree. In the Details area, click the ellipsis (...) next to the Format field.
3. Select the format of the data values from the Format list. Valid options include:
   - As is - The exact value of the column is displayed without any formatting.
   - Currency - Column values are displayed as currency. The locale settings of the current session determine the currency unit that is displayed.
   - Decimal - Column values are displayed as decimals.
   - Scientific - Column values are displayed in scientific notation.
   - Percent - Column values are displayed as percentages.
   - Custom - Data point values are displayed according to the pattern that you specified in the Pattern field.

The following formats are available for numeric data type:

The following formats are available for date data type:

The following formats are available for timestamp data type:

As is - The exact value of the column is displayed without any formatting.
Database - The date is displayed in the format of the database.
Date - Select the date presentation from the available formats.
Custom - The date is displayed according to the pattern that you specified in the Pattern field.

As is - The exact value of the column is displayed without any formatting.
Database - The timestamp is displayed in the format of the database.
Timestamp - Select the timestamp presentation from the available formats.
Date - Only the date component of the timestamp is displayed.
Time - Only the time component of the timestamp is displayed.
- **Custom** - The timestamp is displayed according to the pattern that you specified in the *Pattern* field.

4. To include thousands separators with data point values, select the **Thousands separator** check box.

5. Specify the number of decimal places that are displayed in the *Decimal places* field. A value of '0' indicates that no decimal places are displayed.

6. Select the format of negative numbers in the **Negative numbers** list.

7. Click **OK**. The Choose a Format window closes and control returns to the Create New Quick Report wizard.

**Assigning custom data objects to quick report columns**

You can assign custom data objects that you want to be opened when you click a column in your quick report.

**Before you begin**

You must open the Create New Quick Report wizard.

**About this task**

To specify click events for a column:

**Procedure**

1. Open the **Format** page of the Create New Quick Report wizard.

2. Expand either a grouping or detail column node and select **Click Events** node from the tree. The **Click Events** page of the Create New Quick Report wizard contains two separate pages:
   - **Drill objects** page on which you can assign objects to the selected column.
   - **Column prompts** page on which you can specify variable values that must be passed to the assigned object.

3. On the **Drill objects** page, click **Add new drill object** and browse to the required object in a repository or file system. If the selected object contains prompts, they are displayed in the **Prompts Assignments** list.

   **Note:** Several objects can be assigned to one and the same column. If you click the column, the first object in the list of assigned objects will be opened. To change the order of the objects by clicking **Move Object Up** or **Move Object Down**.

4. You can specify column parameters that will be passed to the assigned object on the **Column prompts** page.

5. To add a column prompt, click **Add new column prompt** and specify its parameters including column prompt name, type, and value. Available type options are:
   - **Constant** to use a constant value
   - **Column** to set the column that will provide the value
   - **Input prompt** to set the column that will provide the value

6. Assign the created column prompts with the required prompts of the data object in the **Prompts Assignments** area on the **Drill objects** page. You can try
to assign prompts automatically by clicking Automatch columns by name. If the names of a column prompt and an object prompt do not coincide, you can assign them manually.

7. If you have specified all the required parameters for the quick report, click Finish to close the Create New Quick Report wizard and display the quick report on the Design page of the editor where you can manually customize the look of the report.

Related tasks:

“Arranging grouping and detail columns” on page 180
You can add and remove the grouping and detail columns from a quick report and convert a grouping column to a detail column and vice versa.

“Specifying how to display the report title and report total information” on page 182
You can specify how to display the report title and report total information of a quick report.

“Specifying how to display column headers” on page 183
You can specify the look of column headers and the order in which they are displayed in a quick report on the Columns Headers page of the Create New Quick Report wizard.

“Creating total and detail column sets” on page 186
You can associate detail columns that you do not always want to be displayed in a quick report with a total column.

“Applying conditional formatting” on page 187
You can use a conditional format in quick reports to help you visually explore and analyze data, discover critical issues, and identify patterns and trends.

“Specifying format options for the Grouping Columns” on page 190
You can specify format options for the Grouping Columns of a quick report.

“Specifying how to display Header and Footer information” on page 191
You can specify the how to display Header and Footer information for a grouping column of a quick report.

“Specifying format options for the Details Columns” on page 192
You can specify how to display the Details Columns of a quick report.

“Specifying quick report column format options” on page 194
You can specify the formatting options of individual columns of your quick report.

Editing quick reports
You can edit the layout of an existing Quick report by adding, removing, or reordering the grouping and detail columns.

About this task

You can add, remove or reorder the grouping and detail columns in the report on the Columns page of the Edit Quick Report wizard. To open the Columns page of the wizard, click the Edit Quick Report Columns toolbar button ( )

You can change the report title and totals line, as well as the formatting options of grouping and detail columns on the Format page of the Edit Quick Report wizard. To open the Format page of the wizard, click the Edit Quick Report Format toolbar button ( ).
These pages are similar to the Column and Format pages of the Create New Quick Report wizard.

You can also modify and change the data source you use to create the quick report.

**Changing data source**
You can change data sources for already created quick report.

**Before you begin**
You must create a quick report.

**About this task**
To change data source:

**Procedure**
1. With a quick report opened in Preview window, select **Report > Change data source**.
2. On the first page of Edit Quick Report wizard, specify a new data source. You can select either a previously saved data source or an opened one.
3. If columns in the new data source match the columns of the previously specified data source, you can click **Finish** to close the wizard and update the report. Otherwise, click **Next** to proceed.
4. On the second page of the wizard, match the data source columns with the columns of the report. If there is no required match to a report column, select the **Remove unmatched columns** check box to proceed. All such columns will be excluded from the report.
5. Click **Finish** to close the wizard and update the report.

**Note:** All report columns remain their initial properties. If after changing data source the column output format does not match the column data type, the default format for this data type is used.

**Related tasks:**

“Editing data source”
You can edit related data sources information for already created quick report.

**Editing data source**
You can edit related data sources information for already created quick report.

**Before you begin**
You must create a quick report.

**About this task**
To edit data source:

**Procedure**
1. With a quick report opened in **Preview** window, select **Report > Edit data source**. The specified data source opens in a separate editor.
2. Edit and save the data source. When you save the modified data source, its columns are compared with the columns of the report. If there is no full compliance, you must match the columns manually in the Edit Quick Report wizard.

3. On the Columns page of the Edit Quick Report wizard, match the data source columns with the columns of the report. If there is no required match to a report column, select the Remove unmatched columns check box to proceed. All unmatched columns will be excluded from the report.

   **Note:** If it is impossible to match the report columns and data source columns, close the Edit Quick Report wizard and edit the data source. If you do not match the columns, data source will not be saved within the report.

4. Click Finish to close the wizard and update the report.

   **Note:** All report columns remain their initial properties. If after editing the column output format does not match the column data type, the default format for this data type is used.

**Related tasks:**

“Changing data source” on page 198

You can change data sources for already created quick report.

### Modifying your quick report design

Once you have created a quick report, you can make simple changes to the layout right in the Preview mode.

**Before you begin**

You must create a quick report.

**About this task**

To modify a quick report in Preview mode:

**Procedure**

1. To expand or collapse the content of a group, click the plus or minus sign to the left of the name of this group.
2. To collapse all groups and show only aggregated information, click the minus sign in the header of the grouping column.
3. To view or hide the contents of detail columns assigned to a total column, click the arrow in the header of the total column.
4. To change the columns order, you can select a column and drag it to another position. The red vertical line indicates the possible column position.
5. To change the width of a column or the height of a row, you can drag the column or row border.
6. To move labels and charts within the header and footer of grouping columns, simply drag them to a new position.

   **Note:** You can also make changes to the size and position of components in the Properties view.

7. To add an extra text field to the header or footer of grouping columns, right-click the header or footer and select Add Text to Break.
8. To modify the titles displayed in the header or footer and the text fields that you add, use the Properties view.

9. To add a chart to the header or footer of grouping columns, right-click the header or footer and select Add Chart.

10. To quickly change the sorting of grouping or detail columns, hide selected columns, or add calculated columns, use the column pop-up menu.

---

**Form menu**

In DataQuant, the Form menu becomes available when a form is the active object in the editor.

The following menu commands are available:

**Data Source Object**

Opens the Data Source Object dialog where you can specify the data source object that will be associated with the form. The data source object for a form is the object that is run to obtain the query results that are formatted into a report.

**Convert to Visual Report**

Converts the active form into a visual report. The generated visual report opens in a new window.

**Convert to HTML**

Opens the Convert to HTML dialog, which contains the following options:

- Specify which tags will be added to the form in The following tags will be added to the form list box.
- Check the Include tabular data as HTML table check box to insert any table data into a HTML formatted table.
- Check the Adjust additional form values for HTML table tags check box to have the application automatically adjust form values for HTML formatting.
- Check the Save as user defaults check box to save the current settings and selections as user defaults.
- Click the User Defaults button to reset to user defaults.
- Click the Product Defaults button to reset to product defaults.
- Click the Clear Values button to clear all selected values from the Value column of the list box.

**Set Data Source**

Select a data source (from a list of available data sources in your repository) where you want the active query to run. The Set Data Source dialog opens.

**Set User Information**

Opens the Set User Information dialog where you specify the user ID and password that DataQuant will use to connect to the data source.

**Set Font**

Opens the Font dialog where you can change the display font attributes of the current classic report in the active window.
Chapter 11. Working with forecasts

Forecasts allow you to predict future values of time series historical data.

With forecasting capabilities, you can make projections of future values based on past values. Using forecasts, organizations can prepare for changes in economic or competitive conditions by analyzing time series historical data to predict performance and future trends. For example, in a supply chain, if the forecast demand matches the actual demand then significant efficiencies can be achieved in terms of production, distribution, and return.

DataQuant forecasts use various predictive methods based on mathematical algorithms that model the future demand based on time series historical data that can be sourced from queries and tables containing date or time columns. The overall objective is to choose a time series method that produces a best fit model of past values, by identifying existing patterns in the data and projecting the model into the future to generate the forecast.

The following methods can be used to forecast future values:

- If the time series is relatively stationary with no overall tendency to fluctuate at one part of the series as compared to another part of the series, then Moving Average, Weighted Moving Average, or Single Exponential Smoothing provide the best fit model.
- If the time series has a trend with a consistent upward or downward movement over time, then Double Exponential Smoothing provide the best fit model.
- If the series has a trend and seasonality with a pattern of peaks and troughs that repeat themselves over a time-frame of usually less than or equal to a year, then Holt-Winters method provide the best fit model.
- If the series has a trend, seasonality and cyclicity with a pattern of peaks and troughs that repeat themselves over an extended time-frame usually greater than a year, then the Multiplicative Decomposition method provide the best fit model.
- If the series displays none of the aforementioned methods, then Neural Networks will be used to mathematically fit the historical data.
- If there are theoretical reasons to indicate that the data should follow a clear mathematical function, then one of the Curve fitting methods can be used.
- In addition to the aforementioned methods, the forecaster is also able to manually adjust any predicted values based on the forecaster's knowledge and any external events.

As most new users discover, the ability to quickly plot and compare each forecast method is a major feature of DataQuant forecasts. However, a forecaster’s knowledge and experience help to reduce the possibilities and consequently provide greater confidence and reliability in the forecast.

Creating forecasts

You can create a forecast using the Forecasts editor. You use the Group and Model pages of the Forecasts editor to set up a forecasting model.
About this task

To create a forecast, perform the following procedure:

Procedure

1. Open the Forecasts editor in one of the following ways:
   - With an active query in the Editor window, select Query > Transfer To > Forecast
   - Select File > New > Other
     The New Wizard wizard opens. From the DataQuant Objects folder select the Forecasts wizard
   - Click the New Forecast toolbar icon.
     The Forecast editor opens in a separate tab.
2. Select the query or table that will be used to source the historical data.
3. Specify the Date Parameters options.
4. Specify the forecasting model’s grouping.
   The Grouping Hierarchy is used to specify what values are grouped and how they are ordered. For example, in wine sales, group by wine type and then location or by location then wine type.
5. Specify the query column that contains the values that will be forecast and the method of aggregation.
6. Specify the forecasting model’s construction strategy, and distribution strategy:
   - The Construction Strategy is used to specify either a top-down or bottom-up approach, where the root node is at the top and the leaf node is at the bottom of a hierarchical tree diagram.
   - The Distribution Strategy specifies how the forecast values are distributed using the top-down construction strategy (e.g., from root to leaf nodes).
7. Specify Forecasting Models options including the forecasting models that are used, and their associated parameters.
8. Click Run Forecast on the toolbar. The graphed forecast is displayed in the Results editor.
9. Experiment with one or more forecasting models to determine the best fit. Validation methods include:
   - Validating the forecasting model using performance measures.
     There are five performance measures that may be used as comparative criteria, including:
     - Cumulative Forecast Error
     - Mean Absolute Deviation
     - Mean Square Error
     - Mean Absolute Percent Error
     - Tracking Signal
     In most cases the Mean Square Error is used as the comparative criteria.
   - Validating the forecasting model time series elements in accordance with any observed trend, seasonality, and cyclicity.
     In the case of trend, an observed trend at a lower node may influence the choice of forecasting model for the entire forecast. For example, when marketing a new product it may be wiser to base the forecasting model on a smaller, more representative demographic with observable trends than use a larger, more diverse demographic with distorted or no observable trends.
10. Save the forecast to a file or to the repository.

Note: You can use saved forecasts as query objects when creating analytical queries, prompt hierarchies, drill down paths, quick reports, and visual projects.

Specifying forecast data source options

You use the Data Source pane of the Group page to specify the data source of a forecast.

About this task

To create a forecast and specify data source options:

Procedure

1. With an active data object that contains at least one date format column in the editor window, select Query > Transfer To > Forecast. The Forecast editor opens. You can choose to embed a query or select a query to link to. An embedded data object is saved independently to the forecast. Any updates made to the original source after the forecast is created are not expressed. A linked data object creates a link between the data source and the saved forecast. Any updates made to the original source are automatically expressed in the forecast.

2. To specify a data object to embed, select Embedded and click Import. The Open window opens.

3. Navigate to and select the data object that you want to embed in the forecast and click Finish. The Open window closes.

4. Click New. A new query opens in the SQL editor. Create the query that you want to embed in the forecast and save it.

5. Click Edit. The embedded object opens in the editor. Edit the object and save it.

6. If you select Linked, click the ellipsis (...) next to the Path field. The Open window opens.

7. Navigate to and select the object that you want to link to the forecast and click Finish. The Open window closes and the path to the query is displayed in the Path field.

Note: Any query that you embed or link to must contain at least one date format column.

Specifying forecast date parameters

You use the Date Parameters pane of the Group page to specify the date range options of the forecast.

About this task

To specify date parameters of the forecast:

Procedure

1. Open the Group page of the Forecast editor.

2. Select the date format column that you want to use to create the forecasted values from the Date column list.
Note: Only date format columns are available in the Date column list. The following formats are available for date:
- YYYYxMMxDD
- MMxDDxYYYY
- DDxDDxYYYY
- YYxDDxDD
- MMxDDxYY
- DDxDDxYY
For time:
- HHxMMxSS
- HHxMM
For timestamp:
- YYYYxMMxDD HHxMMxSS
- DDxDDxYYYY HHxMMxSS
- YYYYxDDxDD HHxMM
- DDxDDxYY HHxMMxSS
- YYxDDxDD HHxMM
- DDxDDxYY HHxMM

3. Select the historical data values that you want to use to calculate the forecasted values from the Historical data list. The available options are:
   a. All data: Select this option to specify that all data values will be used to calculate the forecasted values.
   b. From: Select this option to specify the starting data values that will be used to calculate the forecasted values. Enter the start date in the From calendar field. You can also enter the end date in the To calendar field. In this case, the data values retrieved between the start and end dates will be used to calculate the forecasted values.
   c. Last: Select this option to specify that the data values of the last number of periods will be used to calculate the forecasted values. Specify the number of last periods in the Periods field.

4. Select the time period over which to aggregate data values from the Period list.
5. Specify the number of periods to calculate forecast values in the Number of future periods field.

### Specifying forecast grouping hierarchy options

You use the Grouping Hierarchy pane of the Group page to specify the columns that group the forecast data.

**About this task**

To specify grouping hierarchy options:

**Procedure**

1. Open the Group page of the Forecast editor
2. Select one or more columns from the Available columns list box and click the > button. The column or columns are added to the Selected columns list box.
3. Select one or more columns from the **Selected columns** list box and click the < button. The column or columns are removed from the **Selected columns** list box and returned to the **Available columns** list box.

4. Click the >> or << buttons to add or remove all of the selected columns.

5. To change the order of the selected columns, click **Move Selected Column Up** or **Move Selected Column Down**.

6. Click the Run toolbar button to run the forecast. The high level summary is displayed on the forecast.

**Note:** To see the results of other hierarchy levels, select the level that you want to display on the forecast in the Outline view.

---

**Specifying forecast strategy options**

Use the **Strategy** pane of the **Forecast** editor to specify how the forecasted data is formatted.

**About this task**

To specify strategy options:

**Procedure**

1. Open the **Model** page of the Forecast editor

2. Select the column that contains the values to forecast from the **Forecast column** drop-down list. All available numeric columns are displayed in the **Forecast column** drop-down list.

3. Specify how the forecasted values will be aggregated in the **Aggregation strategy** drop-down list. The aggregation strategy is largely dependent on the type of the value column that you are forecasting. In most cases you would use the **Group sum** option. There are instances however, such as aggregating temperatures, where you would use the **Group average** option.

4. Specify how the forecast model is constructed in the **Construction strategy** drop-down list. If you select **Leaf**, the model is calculated at leaf level and aggregated upwards using a bottom-up approach. This strategy is most effective when there is a full history at the lower reporting level (for example, a rich data set with no nulls at the bottom level). If you select **Root**, the model is calculated at root level and distributed to leaf nodes using a top-down approach. A root level strategy is more effective when there is a smaller data set.

5. If you select the **Root** construction strategy, select how to distribute the root level forecast value to leaf levels in the **Distribution strategy** drop-down list. Available values include:
   - **Equal** - Equally distributes the root level forecasts to each leaf level.
   - **Historical total** - Distributes root value forecasts proportionally to sub-levels according to the total contribution of each group across the whole historical data set.
   - **Most recent period** - Distributes root value forecasts to the most recent forecasted period.

---

**Specifying forecasting models options**

Use the **Forecasting Models** pane of the **Forecasts** editor to specify the forecasting models that are used to calculate the forecasted data.
About this task

To specify forecasting models options:

Procedure

1. Open the Model page of the Forecast editor.
2. Click Add. The Forecast model selection page of the Forecast wizard opens.
3. Select a forecast model. Available forecast models include:
   - Simple moving average
   - Weighted moving average
   - Single exponential smoothing
   - Double exponential smoothing
   - Holt-Winters method
   - Multiplicative decomposition
   - Curve fitting
   - Polynomial regression
   - Neural network
4. Click Next. The parameters page of the specified forecast model opens.
5. Specify the parameters of the forecast model and click Finish. The Forecast wizard closes and the forecast model is displayed in the Forecasting Models table.
6. To delete a forecast model from the Forecasting Models table, click Remove Model.
7. To edit a forecast model parameters, perform the following steps:
   a. Select a forecast model and click Edit Model. The parameters page of the selected forecast model opens.
   b. Edit the parameters of the forecast model and click Finish. The Forecast wizard closes and the changes are saved.
8. To copy a forecast model, click Copy Model. A duplicate model is created and displayed in the Forecasting Model table. This feature is useful if you want to test several models of the same type with slightly different parameters each time.

Setting simple moving average parameters

A moving average smooths a time series by forecasting the value for each period using the average of several previous periods.

About this task

To specify simple moving average parameters:

Procedure

1. Open the Model page of the Forecast editor.
2. Click Add. The Forecast model selection page of the Add Model wizard opens.
3. Click Simple moving average and then click Next. The Simple moving average page of the Add Model wizard opens.
4. Select the number of previous time periods to use to calculate the moving average from the Number of periods drop-down list.
5. Click Finish. The Add Model wizard closes and the forecast model is displayed in the table.

**Setting weighted moving average parameters**

A weighted moving average smooths a time series by forecasting the value for each period using the weighted average of several previous periods.

**About this task**

To specify weighted moving average parameters:

**Procedure**

1. Open the Model page of the Forecast editor.
2. Click Add. The Forecast model selection page of the Add Model wizard opens.
3. Click Weighted moving average and then click Next. The Weighted moving average page of the Add Model wizard opens.
4. Specify the number of previous time periods to use to calculate the moving average in the Number of periods drop-down list.
5. Specify a weight for each of the previous periods in the Weight column of the table. The periods are listed from the earliest to the most recent. For example, giving a high weight to the last period in the list will increase the importance of the most recent data relative to earlier data.
6. Click Finish. The Add Model wizard closes and the forecast model is displayed in the table.

**Setting single exponential smoothing parameters**

A single exponential smoothing is a type of weighted moving average where the weights decrease exponentially for each successive time period.

**About this task**

To specify single exponential smoothing parameters:

**Procedure**

1. Open the Model page of the Forecast editor.
2. Click Add. The Forecast model selection page of the Add Model wizard opens.
3. Click Single exponential smoothing and then click Next. The Single exponential smoothing page of the Add Model wizard opens.
4. Specify a value in the Exponential smoothing constant field. The number must be between zero and one. High values give recent periods more weight and low values give more weight to past data.
5. Click Finish. The Add Model wizard closes and the forecast model is displayed in the table.

**Setting double exponential smoothing parameters**

A double exponential smoothing allows for exponentially decreasing weights that are applied to both previous values and previous trends between values.

**About this task**

To specify double exponential smoothing parameters:
Procedure
1. Open the Model page of the Forecast editor.
2. Click Add. The Forecast model selection page of the Add Model wizard opens.
3. Click Double exponential smoothing and then click Next. The Double exponential smoothing page of the Add Model wizard opens.
4. Specify a value in the Exponential smoothing constant field. The number must be between zero and one. High values give recent periods more weight and low values give more weight to past data.
5. Specify a value in the Trend smoothing constant field. The number must be between zero and one. High values give recent trends more weight and low values give more weight to past trends.
6. Click Finish. The Add Model wizard closes and the forecast model is displayed in the table.

Setting Holt-Winters method parameters
The Holt-Winters method allows for a weighted exponential smoothing that takes seasonality into account.

About this task
To specify Holt-Winters method parameters:

Procedure
1. Open a forecast in the Model editor.
2. Click Add. The Forecast model selection page of the Add Model wizard opens.
3. Click Holt-Winters method and then click Next. The Holt-Winters method page of the Add Model wizard opens.
4. Select the frequency at which you expect to see seasonal effects on the data from the Seasonal frequency drop-down list.
5. Specify a value in the Smoothing constant (alpha) field. The number must be between zero and one. High values give recent periods more weight and low values give more weight to past data.
6. Specify a value in the Seasonal constant (beta) field. The number must be between zero and one. High values give recent periods more weight and low values give more weight to past data.
7. Specify a value in the Trend constant (gamma) field. The number must be between zero and one.
8. Click Finish. The Add Model wizard closes and the forecast model is displayed in the table.

Setting multiplicative decomposition parameters
A multiplicative decomposition extracts trend, seasonality, and cyclicity information from the data and uses these elements to predict future values.

About this task
To specify multiplicative decomposition parameters:

Procedure
1. Open a forecast in the Model editor.
2. Click Add. The Forecast model selection page of the Add Model wizard opens.
3. Click **Multiplicative decomposition** and then click **Next**. The Multiplicative decomposition page of the Add Model wizard opens.

4. Specify the frequency at which you expect to see seasonal effects on the data in the **Seasonal frequency** drop-down list.

5. Specify the frequency at which you expect to see cyclical effects on the data in the **Cycle length** drop-down list.

6. Click **Finish**. The Add Model wizard closes and the forecast model is displayed in the table.

### Setting curve fitting parameters

A curve fitting finds a mathematical function that most closely fits the data. This can be a line of best fit, exponential curve, or logarithmic curve.

**About this task**

To specify curve fitting parameters:

**Procedure**

1. Open a forecast in the **Model** editor.
2. Click **Add**. The Forecast model selection page of the Add Model wizard opens.
3. Click **Curve fitting** and then click **Next**. The Curve fitting page of the Add Model wizard opens.
4. Select a curve type radio button. There are three types of curve types:
   - **linear regression** - Select this option if the rate of change in forecast column values across periods is relatively constant.
   - **power regression** - Select this option if the rate of change in forecast column values across periods is increasing over time.
   - **logarithmic regression** - Select this option if the rate of change in forecast column values across periods is decreasing over time.
5. If you click **power regression**, select the exponential function that you want to use.
6. Click **Finish**. The Add Model wizard closes and the forecast model is displayed in the table.

### Setting polynomial regression parameters

A polynomial regression finds a mathematical function of a certain order that most closely fits the data.

**About this task**

To specify polynomial regression parameters:

**Procedure**

1. Open the **Model** page of the Forecast editor.
2. Click **Add**. The Forecast model selection page of the Add Model wizard opens.
3. Click **Polynomial regression** and then click **Next**. The Polynomial regression page of the Add Model wizard opens.
4. Specify the order of the polynomial in the **Order** drop-down list.
5. Specify how the coefficients are set by clicking one of the radio buttons:
   - Click the first radio button to let the system calculate the coefficients.
b. Click the second radio button to manually set the coefficients.

6. If you click the second radio button, specify the value of each of the coefficients in the Value column of the table.

7. Click Finish. The Add Model wizard closes and the forecast model is displayed in the table.

**Setting neural network parameters**

A neural network uses a portion of the historic data to derive patterns that are then used to predict additional data.

**About this task**

The stage needs to be set just so.

**Procedure**

1. Open the Model page of the Forecast editor.
2. Click Add. The Forecast model selection page of the Add Model wizard opens.
3. Click Neural network and then click Next. The Neural network page of the Add Model wizard opens.
4. Specify a value in the Allowable reduction (%) field.
5. Specify a value in the Allowable increase (%) field.
6. Select the frequency at which you expect to see seasonal effects on the data from the Seasonal frequency drop-down list.
7. Specify how many times the neural network runs the data to train itself in the Training iteration drop-down list.
8. Specify the type of neural network algorithm to be used. Available algorithms include:
   - Gradient descent - A first-order optimization algorithm.
   - Quick prop - A slight variation of the standard backpropagation of error algorithm.
   - R-prop - A resilient backpropagation of error algorithm.
9. Select the number of hidden neurons from the Hidden neurons drop-down list.
10. Specify the random seed of the neural network in the Random seed field.
11. Specify a value in the Training set (%) field.
12. Select the Show training progress check box to display the training progress of the neural network.
13. Click Finish. The Add Model wizard closes and the forecast model is displayed in the table.

---

**Monitoring forecast model performance**

You use forecast performance metrics to determine which model produces the best fit for the forecasted data.

**About this task**

To generate a summary of a forecast model's performance metrics:
Procedure
1. Open the forecast that you want to test.
2. Open a forecast in the Forecast editor.
3. Run the forecast model that you want to test. The forecast results are displayed in the Results editor.
4. Click the Create Performance toolbar button. A table containing the forecast model's performance metrics is displayed.
   **Cumulative Forecast Error**
   Equal to the sum of differences between predicted and actual values.
   **Mean Absolute Deviation**
   Equal to the sum of the absolute values of the forecast error divided by the number of values. This metric tends to provide the best indicator of performance and is used as the default comparison criterion in forecast graphs.
   **Mean Square Error**
   Calculated as the sum (or average) of the squared error values. This performance metric is very sensitive to unique or large values, hence the error is amplified.
   **Mean Absolute Percent Error**
   Calculated as a percentage of the absolute difference between predicted and actual values divided by the number of values.
   **Tracking Signal**
   Calculated as a ratio of cumulative forecast error to the mean absolute deviation.
   In general, the closer the error is to zero, the better the performance of the model (for example, a performance error equal to zero implies a perfect fit between the predicted and actual values).
5. Optional: If you want to view the forecast hierarchical structure, open the Outline view by selecting Window > Show view > Outline from the main menu.

Observing trends
You use trends to track in what direction the forecasted data moves.

About this task
To observe the trend of the historical data:

Procedure
1. Open the forecast for which you want to create a trend.
2. Open a forecast in the Forecast editor.
3. Run the forecast model that you want to test. The forecast results are displayed in the Results editor.
4. Click the Create Trend toolbar button. A trend graph and a trend table are displayed in the Results editor.
5. Observe the three types of trends in the trend chart to determine if there is a significant trend to the data, and in what direction it moves.
Local Trend
The difference between the current value within a period and the value for the last period.

Global Trend
The difference between the current value within a period and the average of all values. The period, order, and magnitude of fluctuation between positive and negative values can be used to determine the significance of the linear trend.

Linear Trend
The linear regression value for the current period, as displayed in the trend graph.

6. Optional: If you want to view the forecast hierarchical structure, open the Outline view by selecting Window > Show view > Outline from the main menu.

Observing seasonality
Seasonality can be observed in a graph as a repeated pattern of regularly spaced peaks and troughs that have a consistent direction and approximately the same magnitude relative to the trend.

About this task
Some examples of seasonality might be a sharp escalation in retail sales in response to holiday shopping, increase in water consumption in the summer due to warmer weather, and increase in government spending at the end of the fiscal year. To observe seasonality within the historical data:

Procedure
1. Open the forecast for which you want to observe seasonality.
2. Open a forecast in the Forecast editor.
3. Run the forecast model that you want to test. The forecast results are displayed in the Results editor.
4. Click the Create Seasonality toolbar button . The Create Seasonality window opens.
5. Select the seasonality time period from the Time period drop-down list.
6. Select the historical data averaging method from the Seasonality type drop-down list.
7. Click OK. The Create Seasonality window closes and a seasonality chart and seasonality graph are displayed in the Results editor. The seasonality table displays, for each period, the overall average for all seasons and the average for each seasonality period.
8. Optional: If you want to view the forecast hierarchical structure, open the Outline view by selecting Window > Show view > Outline from the main menu.

Observeing cyclicity
Cyclicity allows you to observe periodic fluctuations that repeat within a value's time series, usually over a time period that is greater than one year.
About this task

Some examples of industries that experience cyclical demand are automobile, defense, mineral resources, and construction industries. To observe cyclicity within historical data:

Procedure
1. Open the forecast for which you want to observe cyclicity.
2. Open a forecast in the Forecast editor.
3. Run the forecast model that you want to test. The forecast results are displayed in the Results editor.
4. Click the Create Cyclicity toolbar button. The Create Cyclicity window opens.
5. Select the cycle length from the Number of points drop-down list. The cycle length is the number of periods expected in a complete cycle.
6. Click OK. The Create Cyclicity window closes and a cyclicity chart and cyclicity table are displayed in the Results editor. The cyclicity table displays, for each period in a cycle, the overall average for all cycles and the average for each individual cycle. For example, if a cycle has twelve periods and the data history is three years then average values for three cycles are displayed.
7. Optional: If you want to view the forecast hierarchical structure, open the Outline view by selecting Window > Show view > Outline from the main menu.

Editing chart properties

You use the Chart Properties wizard to edit the properties of forecast result charts.

About this task

Once you run a forecast model, you can customize the generated chart formatting to your specifications. To edit a forecast result chart:

Procedure
1. Open a forecast in the Forecast editor.
2. Run the forecast model that you want to test. The forecast results are displayed in the Results editor.
3. Click the Chart Properties icon in the menu bar. The Chart Properties wizard opens.
4. Edit the chart formatting options and click OK. The Chart Properties wizard closes and the changes are saved.

Editing grid properties

You use the Grid Properties wizard to edit the properties of forecast result grids.

About this task

Once you run a forecast model, you can customize the generated grid formatting to your specifications. To edit a forecast result grid:
Procedure
1. Open a forecast in the Forecast editor.
2. Run the forecast model that you want to test. The forecast results are displayed in the Results editor.
3. Click the Grid Properties toolbar button. The Grid Properties wizard opens.
4. Select Columns in the tree. The Columns options are displayed.
5. Specify the columns to display in the grid by selecting the check boxes in the Visible column.
6. Select a Value Column in the tree. The Value Column options are displayed.
7. Specify the column header name in the Text field.
8. Specify the font of the header name in the Font field.
9. Specify the background color of the column header cell in the Background field.
10. Specify the font of the text in the detail cells in the Font field.
11. Specify the format of the text in the detail cells in the Format field.
12. Specify the background color of the detail cells in the Background field.
13. Repeat the previous steps for each of the value columns.
14. Click OK. The Grid Properties wizard closes and the changes are saved.

Exporting forecasts
You can export charts and query result sets of generated forecasts to a file system or database using the Export Chart or Export Query Results wizard.

About this task
- To export a chart see the “Exporting charts” on page 230 topic.
- To export query results to a database see the “Exporting query results to a database” on page 118 topic.
- To export query results to a file see the “Exporting query results to a file” on page 119 topic.

Specifying print options for a forecast
You can specify the page setup options that will be used when you print a forecast opened in the editor window.

About this task
To control the appearance of printed pages for a forecast:

Procedure
1. With an active object in the editor window, open the Page Setup window by selecting File > Page Setup.
2. Select the printer that you want to use from the Printer list.
3. For detailed settings click Properties. The Page Setup window opens.
4. Select the size of the printable area from the Size list.
5. Select the paper tray to use from the Source list.
6. Select the orientation of the printout from the Orientation area.
7. Specify the size of the margins of the page in the Margins fields.
8. Click OK to close Page Setup window.
9. In the Page area, specify the forecast objects that you want to print by selecting or clearing the Chart and Table check boxes.

10. In the same area, select one of the available page options:
   a. **Print on separate pages** to print the forecast objects, the chart and the table, on separate pages.
   b. **Print on the same page** to print both forecast objects on one page.

11. In the Scale Options area, select one from the available printing options:
    a. **Fit to printable area** to fit the forecast size to the printable area.
    b. **Scale print to %** to specify the required forecast size in percent.

12. To make the specified values default, click **Set as Default**.

13. Click **OK** to close Page Setup window. The options that you specified are used to control the appearance of the printed object.
Chapter 12. Working with charts

You can use charts for graphical representation of the data that is retrieved from queries.

About this task

For QMF queries you can create only one chart for a single result set. For visual queries you can create several different charts for a single result set.

The following topics describe how to create and customize charts:

Creating charts

You can create a chart from a query result set with the help of the New Chart wizard.

Before you begin

You must run a query and have a query result set open on the Results tab.

About this task

To create a chart, do the following actions:

Procedure

1. Open the New Chart wizard in one of the following ways:
   a. From the main menu, select Results > Display Chart and specify the type of the chart that you want to create. Then go to the step 3 of the procedure.
   b. Click the Display Chart toolbar button.

2. On the Chart type page, specify the type of the chart. For more information about chart types, see “Specifying chart type options” on page 218 topic. Click Next.

3. On the Data series page, specify which columns of a result set are used to generate data points of a chart. For more information, see “Specifying data series options” on page 218 topic. Click Next.

4. On the Format page, specify the options to define the visual output of the chart. For more information, see “Specifying format options” on page 219 topic.

5. Click Finish to close the New Chart wizard and display the created chart. If you create a chart from QMF query results, it is displayed on the Chart tab. If you create a chart from visual query results, it is displayed on the Results tab.

6. From the main menu, select Chart > Remove chart to remove a chart that you created from a QMF query result set. To remove a chart that you created from visual query results, see the “Managing display modes” on page 228 topic.
Specifying chart type options

You can specify how to display a chart on the Chart type page of the New Chart wizard. Chart type options determine the type of the chart and the visual format of the display mode.

About this task

To specify chart type options:

Procedure

1. From the Chart type list, select the required display mode for the chart. Available options are the following:
   - **Column** – Data points are represented by vertical columns.
   - **Bar** – Data points are represented by horizontal columns.
   - **Line** – Data points are plotted on the X-Y axes using interconnected lines.
   - **Area** – Data points are plotted similar to the line chart but with the area between the data series and x-axis filled with a specified color.
   - **Pie** – Data points are displayed as wedges of a pie.
   - **Scatter** – Data points are plotted as discrete points, using a user-configurable data symbol.
   - **Bubble** – Data points are plotted using spherical data symbols, each of which can be sized according to an expression or column value.
   - **Speedo** – Data points are displayed as points on a dial.
   You can also select Grid mode to display data point in tabular grid.

2. From the Dimension list select the required dimension. Available variants are the following:
   - **2D** – The chart is displayed as a flat, two dimensional objects. It is available for all chart types.
   - **2D With Depth** – The chart is displayed as a two dimensional object, with data points displayed in three dimensions. It is available for Column, Bar, Area, and Pie chart types.

   **Note:** Stacked Area Charts do not support **2D With Depth** dimension.

3. If you select Column, Bar, or Line chart type, select the required subtype. Available options are the following:
   - **Side-by-Side**
   - **Stacked**
   - **Percent Stacked**

Specifying data series options

You can specify which columns of a result set are used to generate data points of a chart on the Data series page of the New Chart wizard.

About this task

To specify the data series options of a chart:
Procedure

1. To specify the column that will populate the category axis, select a column from the Result set tree and click Add to category. The column is added to the Category Axis node of the Series tree.

2. To specify a column that will be displayed as a value, select a column from the Result set tree and click Add to value. The column is added to the Values node of the Series tree.

3. Select aggregation type for value column from the Aggregation drop-down list. Valid options are:
   - No aggregation
   - Count
   - First
   - Last
   - Maximum
   - Minimum
   - Sum
   - Average
   - Standard Deviation

4. Specify the way of aggregating a value column by selecting or clearing the Calculate as formula check box. The Calculate as formula check box is available only if you use a calculated column as a value. The expression you have specified for the calculated column is used as aggregation formula.
   
   If you select the Calculate as formula check box, the fields of the columns that provide data for calculated column (and thus for value column) are aggregated at first. The formula uses already aggregated field values as arguments to calculate the result value.

   If you clear the Calculate as formula check box, the defined formula calculates the value column fields separately. After the calculations the fields are aggregated to calculate the result value.

5. To remove a column from the series tree, select the column and click Remove. The column is added back to the Result set tree.

Specifying format options

Specify Chart Format options to determine the visual formatting of the chart object.

About this task

On the Format page of the New Chart wizard you can specify Series options and Chart Area options.

Series options determine which category axis and value axis series are displayed in the chart, and how they are formatted.

Chart Area options determine the visual formatting of the chart object, such as the chart title, legend, and chart area color schemes.

Specifying Series chart format options

Series options determine which category axis and value axis series are displayed in the chart, and how they are formatted.
About this task

To specify or edit Series chart format options:

Procedure

1. On the Format page of the New Chart wizard, select Series in the tree. The Series options are displayed.
2. Specify whether the chart data points will be colored according to the value series or the category in the Color By list.
3. To include or exclude one of the value series in the chart, select or clear the check box in the Visible column.

Specifying Category chart format options

Category chart format options control how the chart data points are displayed along the category axis.

About this task

To specify Category chart format options:

Procedure

1. On the Format page of the New Chart wizard, select Category in the tree. The Category options are displayed.
2. Specify what data series is used to sort the data points along the category axis in the Order By list.
3. If you specified Values in the Order By list, select the value column that you want to use.
4. Specify whether the data points along the category axis are sorted ascending or descending in the Order list.
5. Select what type of cutoff that is used to restrict the display of data points along the category axis in the Type list. Valid options include:
   • No Cutoff - No cutoff is used to restrict the category axis data points.
   • Item Number - Only the specified number of data points are displayed.
   • Data Value(Less) - Only rows that contain values that are larger than the specified value are displayed.
   • Data Value(More) - Only rows that contain values that are smaller than the specified value are displayed.
   • Percentage(Less) - Only rows that contain values that are more than the specified percentage of the data range.
   • Percentage(More) - Only rows that contain values that are less than the specified percentage of the data range.
6. Specify the value column that you want to use to determine cutoff in the Value Series list.
7. Specify the value that you want to use to determine the cutoff in the Value field.
8. Select the Visible check box to make excluded data points visible.
9. Specify what label is displayed with the excluded data points in the Label field.

Specifying Value(Y)Series chart format options

Value(Y)Series chart format options control how the value series axis is displayed.
About this task

To specify Value(Y)Series chart format options:

Procedure

1. On the Format page of the New Chart wizard, select Value(Y)Series in the tree. The Value(Y)Series options are displayed.
2. Specify the title of the value series axis in the Title field.
3. To make the title of the value series axis visible, select the Visible check box.
4. Specify the color of the value series data points in the Color field.
5. Expand the Value(Y)Series node and select Labels.
6. Specify the labels options of the values series axis.

Related tasks:

“Specifying Labels chart format options” on page 223

Labels chart format options control whether the labels of an axis are displayed and how they are formatted.

Specifying Chart Area chart format options

Chart Area chart format options control how the area of the chart is displayed.

About this task

To specify Chart Area chart format options:

Procedure

1. On the Format page of the New Chart wizard, select Chart Area in the tree. The Chart Area options are displayed.
2. Specify the background color of the chart area in the Background field.
3. To display an outline around the chart area, select the Visible check box.
4. If you are editing the chart properties for a pie chart, select the Auto check box in the Coverage area to automatically make space for multiple pie charts. The Coverage property of a pie chart controls how much of the proportion of the chart area individual pie charts inhabit. The Auto check box ensures that none of the individual pie charts in the chart area overlap.
5. If you clear the Auto check box, specify what proportion of the chart area that the pie chart takes up in the Value field. For example, if you specify a value of 90 in the Value field, the pie chart will inhabit ninety percent of the total height of the chart area.

Specifying Axes chart format options

Axes chart format options control what axes of the chart are included, and how they are displayed.

About this task

To specify Axes chart format options:

Procedure

1. On the Format page of the New Chart wizard, select Axes in the tree. The Axes options are displayed.
2. To include an axis in the chart, select the check box in the Visible column.
3. Specify the title of an axis in the Title column.
4. Select whether an axis will be linear or logarithmic in the **Type** column.
5. Expand the **Axes** node and select **X-Axis**.
6. Specify the **X-Axis** chart format options.
7. Specify the **Y-Axis-1** chart format options.
8. Specify the **Y-Axis-2** chart format options.

**Specifying X-Axis chart format options**

X-Axis chart format options control how the x-axis of the chart is displayed.

**About this task**

To specify **X-Axis** chart format options:

**Procedure**

1. On the **Format** page of the New Chart wizard, expand the **Axes** node, and select **X-Axis** in the tree. The **X-Axis** options are displayed.
2. To display the x-axis of the chart, select the **Visible** check box.
3. To display the title of the x-axis, select the **Visible** check box next to the **Title** field.
4. Specify the title of the x-axis in the **Title** field.
5. Specify the font of the title in the **Font** field.
6. Specify the color of the x-axis in the **Color** field.
7. Select the position of the x-axis from the **Origin** list. Valid options are:
   - **Max** - The x-axis is located above the maximum value of the y-axis.
   - **Min** - The x-axis is located below the minimum value of the y-axis.
   - **Value** - The x-axis is flush against the bottom of the client area of the chart.
8. To display the values of the x-axis according to the category axis, select the **Is category axis** check box. When this check box is selected, data points are displayed evenly along the x-axis, with every major tick mark representing the next data point down the line. When this check box is cleared, data points are displayed unevenly along the x-axis, with major tick marks placed at regular intervals.

**Specifying Y-Axes chart format options**

Y-Axis chart format options control how the y-axis of the chart is displayed.

**About this task**

To specify **Y-Axis** chart format options:

**Procedure**

1. On the **Format** page of the New Chart wizard, expand the **Axes** node, and select **Y-Axis** in the tree. The **Y-Axis** options are displayed.
2. To display the y-axis of the chart, select the **Visible** check box.
3. To display the title of the y-axis, select the **Visible** check box next to the **Title** field.
4. Specify the title of the y-axis in the **Title** field.
5. Specify the font of the title in the **Font** field.
6. Specify the color of the y-axis in the **Color** field.
7. Specify the position of the y-axis in the **Origin** list. Valid options are:
- **Max** - The y-axis is located after the maximum value of the x-axis.
- **Min** - The y-axis is located before the minimum value of the x-axis.
- **Value** - The y-axis is flush against the side of the client area of the chart.

8. Specify whether the y-axis values will be displayed as a linear function or a logarithmic function in the **Type** list.

### Specifying Gridlines chart format options

Gridlines chart options control whether gridlines are displayed and how they are formatted.

**About this task**

To specify **Gridlines** chart format options:

**Procedure**

1. On the **Format** page of the New Chart wizard, expand the **Axes** node.
2. Expand the **X-Axis** or **Y-Axis** node and select **Gridlines** in the tree. The **Gridlines** options are displayed.
3. To display the major gridlines, select the **Visible** check box in the **Major Grid** section.
4. Specify the color of the major gridlines in the **Color** field.
5. To display the minor gridlines, select the **Visible** check box in the **Minor Grid** section.
6. Specify the color of the minor gridlines in the **Color** field.

### Specifying Labels chart format options

Labels chart format options control whether the labels of an axis are displayed and how they are formatted.

**About this task**

To specify **Labels** chart format options:

**Procedure**

1. On the **Format** page of the New Chart wizard, expand the **Axes** node.
2. Expand the **X-Axis** or **Y-Axis** node and select **Labels** in the tree. The **Labels** options are displayed.
3. To display the labels of an axis, select the **Visible** check box.
4. Specify the font of the axis labels in the **Font** field.
5. Click the ellipsis (...) next to the **Format** field. The Choose a format window opens.
6. Select the format of the column cell values from the **Format** list.

   The following formats are available for numeric data type:
   - **As is** - The exact value of the column is displayed without any formatting.
   - **Currency** - Column values are displayed as currency. The locale settings of the current session determine the currency unit that is displayed.
   - **Decimal** - Column values are displayed as decimals.
   - **Scientific** - Column values are displayed in scientific notation.
   - **Percent** - Column values are displayed as percentages.
   - **Custom** - Data point values are displayed according to the pattern that you specified in the **Pattern** field.
The following formats are available for date data type:

- **As is** - The exact value of the column is displayed without any formatting.
- **Database** - The date is displayed in the format of the database.
- **Date** - Select the date presentation from the available formats.
- **Custom** - The date is displayed according to the pattern that you specified in the **Pattern** field.

The following formats are available for timestamp data type:

- **As is** - The exact value of the column is displayed without any formatting.
- **Database** - The timestamp is displayed in the format of the database.
- **Timestamp** - Select the timestamp presentation from the available formats.
- **Date** - Only the date component of the timestamp is displayed.
- **Time** - Only the time component of the timestamp is displayed.
- **Custom** - The timestamp is displayed according to the pattern that you specified in the **Pattern** field.

7. To include thousands separators with column values, select the **Thousands separator** check box.

8. Specify the number of decimal places that are displayed in the **Decimal places** field. A value of '0' indicates that no decimal places are displayed.

9. Click **OK** to close the Choose a format dialog.

10. Specify the angle of the labels of the axis in the **Rotation** field. A value of '0' indicates no rotation. Labels are displayed with no angle. Higher numbers indicate higher angles. Values can be both positive and negative. Positive numbers turn labels counter clockwise and negative values turn labels clockwise.

**Specifying Range chart format options**

Range options allow you to manually format the range and step count of an axis of a chart.

**About this task**

To specify **Range** chart format options:

**Procedure**

1. On the **Format** page of the New Chart wizard, expand the **Axes** node.
2. Expand the **X-Axis** or **Y-Axis** node and select **Range** in the tree. The **Range** options are displayed.
3. To manually set the range of an axis, select the **Is manual Range** check box.
4. To specify the number of steps on the axis, click **Step count** and specify the number of steps in the field.
5. To specify the size of each step on the axis, click **Step size** and specify the size of each step in the text field.
6. Specify the minimum value of the range of the axis in the **Minimum** field.
7. Specify the maximum value of the range of the axis in the **Maximum** field.

**Specifying Title chart format options**

Title chart format options control whether the title of the chart is displayed and how it is formatted.
About this task

To specify **Title** chart format options:

**Procedure**
1. On the **Format** page of the New Chart wizard, select **Title** in the tree. The **Title** options are displayed.
2. To display the title of the chart, select the **Visible** check box.
3. From the **Type** drop-down list, select whether the chart title is automatically generated using the axes columns or is custom text that you enter.
4. Specify the title of the chart in the **Text** field.
5. Specify the font of the title in the **Font** field.
6. Select the location of the chart from the **Location** list.

**Specifying Legend chart format options**
Legend chart format options control whether a legend is displayed and how it is formatted.

About this task

To specify **Legend** chart format options:

**Procedure**
1. On the **Format** page of the New Chart wizard, select **Legend** in the tree. The **Legend** options are displayed.
2. To display the legend of the chart, select the **Visible** check box.
3. To display the title of the legend, select the **Visible** check box that is next to the **Title** field.
4. Specify the title of the legend in the **Text** field.
5. Specify the font of the title in the **Font** field.
6. Specify the position of the legend in the **Position** drop-down list.
7. Specify the font of the text of the legend in the **Font Text** field.
8. To display an outline around the legend, select the **Visible** check box that is next to the **Outline** field.

**Specifying Plot Area chart format options**
Plot Area chart format options control the look and feel of the area of the chart that contains the data axes and the client area.

About this task

To specify **Plot Area** chart format options:

**Procedure**
1. On the **Format** page of the New Chart wizard, select **Plot Area** in the tree. The **Plot Area** options are displayed.
2. Specify the background color of the plot area in the **Background** field.
3. To display an outline around the plot area, select the **Visible** check box that is next to the **Outline** field.
4. Specify the distance that you would like to inset the chart area from the edge of the plot area in the **Insets(points)** fields.
Specifying Client Area chart format options

Client Area chart format options control how the client area of the chart is displayed.

About this task

To specify Client Area chart format options:

Procedure

1. On the Format page of the New Chart wizard, select Client Area in the tree. The Client Area options are displayed.
2. Specify the background color of the client area in the Background field.
3. To display an outline around the client area, select the Visible check box that is next to the Outline field.

Creating charts from a query results grid

You can create a ready-made chart by selecting the required columns from a query results grid. This allows you to review the returned data and decide what to display and how to display it.

Before you begin

You must run a query or visual query and have a query results grid open on the Results tab.

About this task

To create a chart from a query results grid:

Procedure

1. Select the columns that you want to include in the chart by holding the Ctrl key and clicking each column.
2. Click the down arrow next to the Display Chart toolbar button and select the type of chart that you want to display. The Data series page of the New Chart wizard opens. The columns that you selected are displayed in the Series tree.

   Note: Columns are automatically organized into the Category Axis and Values nodes depending on their data types. Category columns are the columns that contain character data, and value columns are the columns that contain numerical data. For example, if you selected a column of department names and a column of yearly salary totals, the department name column would automatically be placed under Category Axis, and the salary column would automatically be placed under Values.

3. Select each value column and select how it is aggregated from the Aggregation list.
5. Specify the different formatting options of the chart and click Finish to close the New Chart wizard and display the created chart.

   If you create a chart from QMF query results, it is displayed on the Chart tab.
   If you create a chart from visual query results, it is displayed on the Results tab.
Work with charts in the Display editor

You can create charts for a visual query result set and manage them using the Display editor.

About this task

All charts that you create for a visual query result set are stored with this result set. You use the Display editor to manage the display modes of a query result set. The Display editor contains the following panes:

- Result set
- Layout structure
- Display modes

The Result set pane displays all available result set columns regardless of the selected display mode. Query result columns are automatically grouped into three types: Category columns, Value columns, and Columns with aggregates. Category columns are columns that contain character data, and it is suggested that they are used for non-numerical data axes, such as the x-axis of a column chart. Value columns are columns that contain numerical data, and it is suggested that they are used for numerical data axes, such as the pie wedges of a pie chart. Columns with aggregates are copies of any columns to which the aggregations are applied. You can use the Result set pane to add groupings and aggregations to any column in the result set.

The Layout structure pane displays the specific layout structure of the selected display mode. Unlike the Result set pane, the Layout structure pane displays only the columns, groupings, and aggregations of the selected display mode. You can use the Layout structure pane to edit the individual layout structures the display modes.

The Display modes pane displays a list of available display modes. Each display mode has an independent layout structure and display settings. You can use the Display modes pane to add, delete, and edit the display modes of the given query result set.

Creating charts in the Display editor

You can create a ready-made chart for a visual query result set by selecting the columns in the Display editor.

Before you begin

You must run a visual query and switch to the Display tab.

About this task

To create a chart in the Display editor, do the following actions:

Procedure

1. In the Result set pane select the columns that you want to include in the chart by holding the Ctrl key and clicking each column.
2. With the mouse button pressed, drag the columns to the Chart tree in the Display modes pane. Release the mouse button. A new chart appears in the Chart tree. This chart becomes default.
3. You can modify the created chart. For more information, see the Editing charts topic.

4. Click the Run Query toolbar button to display the created chart on the Results tab.

Results

The chart that you created for a visual query result set is stored with this result set.

Managing display modes

You use the Display modes pane of the Display editor to add, edit, and delete the display modes of a query result set.

About this task

With the display modes you can visualize result set data in many different ways. You can create any number of display modes with different content and visual formatting. Every result set starts with a grid called Results grid. This is the standard query results grid that is displayed when you run a query for the first time. You can not edit this grid, but you can add a new one to the Grid node of the Display modes tree, and edit its layout as needed.

To manage the display modes of a query result set:

Procedure
1. Navigate to the query that has the result set that you want to edit and open it in the Visual Query editor.
2. Click the Display tab. The Display editor opens.
3. To add a display mode, right-click either Grid or Chart, select New, and select the display mode that you want to add.
4. To rename a display mode, right-click it and select Rename from the pop-up menu.
5. To edit a display mode, right-click it and select Edit from the pop-up menu.
6. To delete a display mode, right-click it and select Remove from the pop-up menu.
7. To move a display mode up or down in the list, right-click it and select Move Up or Move Down from the pop-up menu.
8. To set a given display mode as default, right-click it and select Make Default from the pop-up menu. The default display mode will open first every time the query is run.

Editing query result set columns

You use the Result set pane of the Display editor to modify the columns of a result set.

About this task

To modify result set columns:

Procedure
1. Navigate to the query that has the result set that you want to edit and open it in the Visual Query editor.
2. Click the **Display** tab. The **Display** editor opens.

3. To roll up a column into the top group, right-click the column and select **Top Group** from the menu.

4. To roll up a column into the side group, right-click the column and select **Side Group** from the menu.

5. To add aggregation to a column, right-click the column and select an aggregation type from the menu. Available aggregations are:

   - Count
   - First
   - Last
   - Maximum
   - Minimum
   - Sum
   - Average
   - Standard Deviation

6. To remove aggregation from a column, right-click the column and select **No Aggregation** from the menu.

7. To add a count column to the layout structure of the selected display mode, right-click the (count) column in the **Columns with Aggregates** node and select to **Layout structure** from the menu.

**Editing display mode layout structures**

You use the **Layout structure** pane of the **Display** editor to edit the layout structure of individual display modes.

**About this task**

To modify the layout structure of your query result set:

**Procedure**

1. Navigate to the query that has the result set that you want to edit and open it in the **Visual Query** editor.

2. Click the **Display** tab. The **Display** editor opens.

3. Select the display mode that you want to edit. The layout structure of the selected display mode opens in the **Layout structure** pane.

4. To roll up a column into the top group or side group, click and drag the column to the **Top Groups** or **Side Groups** nodes.

5. To add aggregation to a column, right-click the column and select an aggregation type from the menu. Available aggregations are:

   - Count
   - First
   - Last
   - Maximum
   - Minimum
   - Sum
   - Average
   - Standard Deviation

6. To remove aggregation from a column, right-click the column and select **No Aggregation** from the context menu.
7. To remove a column from the layout structure, right-click the column and select **Remove** from the context menu.

8. To change the order of the columns, click a drag the columns to place them in the order that you want them to appear in the result set.

9. If you are editing the layout structure of a chart, you can click and drag columns to move them to and from the **Category Axis** and **Values** nodes.

---

### Editing charts

You can edit charts and change their visual appearance by modifying the chart properties.

#### About this task

To edit a chart, perform the following procedure:

**Procedure**

1. Open the "Edit chart properties" wizard in one of the following ways:
   a. If you want to edit a chart that is created from a QMF query result set, open a chart on the **Chart** tab and select **Chart > Chart properties** from the main menu.
   b. If you want to edit a chart that is created from a visual query result set, use one of the following options:
      - Open a chart on the **Results** tab and select **Results > Chart properties** from the main menu.
      - In the **Display modes** pane, right-click a chart that you want to edit and select **Edit** from the menu.

2. In the "Edit chart properties" wizard, switch to the appropriate tab to edit chart type, data series, or format options. For more information about chart properties, see the following topics:
   - **Specifying chart type options**
   - **Specifying data series options**
   - **Specifying format options**

3. Click **OK** to save changes and close the "Edit chart properties" wizard.

---

### Exporting charts

You can export charts that are created from query result sets to a file system using the "Export chart" wizard.

#### Before you begin

Open a chart that you want to export on the **Chart** tab, if a chart is created from QMF query results, or on the **Results** tab, if a chart is created from visual query results.

#### About this task

To export a chart:

**Procedure**

1. Open the "Export chart" wizard in one of the following ways:
• With a chart opened in the editor, select **File > Export** to open the Select page of the Export wizard, and then select **Chart** from the list of available export wizards.

• With a chart opened in the editor, click the **Export (→)** toolbar button.

2. Select the source object from the **Sources** list.

3. From the **Export Type** list select the type of the exported file. The following formats are available:
   • PDF
   • PNG

4. Click **Next**. The second page of the "Export chart" wizard opens.

5. In the **Select export parameters** section, specify the size of the image.

6. Click **Update Chart** to refresh the **Chart export preview** window.

7. Click **Finish** to

---

**Specifying print options for charts**

For charts you can specify the page setup options for printing.

**Before you begin**

If a chart is created from QMF query results, open it on the **Chart** tab. If a chart is created from visual query results, open it on the **Results** tab.

**About this task**

To control the appearance of printed pages for a chart:

**Procedure**

1. From the main menu, select **File > Page Setup** to open the Page Setup window.

2. Select the printer that you want to use from the **Printer** list.

3. For detailed settings click **Properties**.

4. Select the size of the printable area from the **Size** list.

5. Select the paper tray to use from the **Source** list.

6. Select the orientation of the printout from the **Orientation** area.

7. Specify the size of the margins of the page in the **Margins** fields and click **OK**.

8. In the **Scale Options** area, select one of the available printing options:
   a. **Fit to printable area** to fit the chart size to the printable area.
   b. **Scale print to %** to specify the required chart size in percent.

9. To make the specified values default, click **Set as Default**.

10. Click **OK** to close the Page Setup window. The options that you specified are used to control the appearance of the printed object.
Chapter 13. Working with visual projects

Visual projects include visual reports and visual dashboards.

About this task

Visual reports

Visual reports are page-based, printable reports that include both formatted text and graphics to display persistent data to a wide variety of users. Visual reports can also contain data driven graphics (such as maps and charts) inserted in different sections (such as the headers or footers) of the report. Each of the data driven graphics can present data from multiple queries that are run across the enterprise.

Ad hoc reports

Ad hoc reports provide a flexible and user-friendly approach to creating reports, while making the overall design process more efficient. They include a scene with a canvas on which you create a report. A canvas can contain data objects such as charts, queries, and dynamarts.

Visual dashboards

Visual dashboards present interactive or persistent data obtained from querying multiple heterogeneous data sources across the enterprise. Visual dashboards present data in a scene format using a wide variety of graphics including charts, graphs, maps and user interface widgets. Data driven graphical objects can be easily linked so that user selections will trigger unique data displays. Content developers can create a visual dashboard that can be viewed by multiple users with either DataQuant for Workstation or DataQuant for WebSphere.

Visual applications

You cannot create visual applications in DataQuant for Workstation starting from Version 2.1 Fix Pack 4. In current release, you can run visual applications that were created in previous versions of DataQuant. The existing visual applications are automatically converted to visual dashboards, if you open them for editing. After conversion the initial visual application remains unchanged. You can edit the created visual dashboard, and then save it to a repository or file. In this case, the created visual dashboard cannot be run in previous versions of DataQuant.

The following functionality specific to visual applications can be now used in visual dashboards:

- Canvas object
- Tree object
- Show dialog event action
- Close dialog event action
- The ability to create folders to store visual dashboard scenes and queries.
Visual project templates

Templates provide standardized, pre-formatted examples of visual projects (visual reports and dashboards). They are reusable components that facilitate the design process.

Visual project designers might use templates for a visual project as the entry-point into their project development process.

Templates help developers avoid repetitive development and promote effective reuse strategies. For example, if an enterprise has multiple visual dashboards that connect to a particular data source, rather than defining the connection properties in multiple visual dashboards, you could create a single template with the required connection. The new visual dashboards that you create can be based on this one template.

There are three template types for visual projects:
- Predefined
- Local
- Repository

Predefined templates

Predefined templates are included with DataQuant for Workstation.

The layout of predefined templates cannot be changed.

Currently there is a single predefined default template included with DataQuant for Workstation. The name of the default predefined template is Blank. The Blank template contains no structural elements.

Predefined templates provide a base-level functionality for a yet to be built business function. For example, you can use a predefined template as the basis for a connecting to a data source, running a standard set of queries or a customized set of charts and tables, connected with lists or other data driven controls.

Local templates

Local templates are those that have been created for your organization and saved locally.

Each template contains predefined content (such as company logos, standardized text, frequent graphical layouts).

The templates are listed by category for organizational purposes.

Typically, local templates are used by individual developers for their own purposes. Visual projects saved as local templates can not be shared via repository services.

Repository templates

Repository templates are those that have been created for your organization and saved in a shared repository.
Each template contains predefined content (such as company logos, standardized text, frequent graphical layouts).

The templates are listed by category for organizational purposes.

Users can edit the structural content of repository templates to suit their needs.

Typically, repository templates are used by multiple developers. Visual projects saved as repository templates are made available to multiple developers through repository services.

Creating template categories for visual projects

Use the Create New [objectname] Template Category wizard to create a template category for your visual reports and dashboards.

About this task

To create a template category for a visual project (visual report or dashboard), do the following steps

Procedure

1. From the main menu, select File > New > Other > DataQuant Objects > [objectname] Template Category and click Next to open the Create New [objectname] Template Category wizard.
2. Specify a name for the template category in the Name field.
3. Optionally enter a comment to describe the template category in the Comment field.
4. Click Finish to close the wizard. The template category that you created for a visual project is added to the appropriate [objectname] Templates folder under the Templates node in the Repository Explorer view.

Note:

The visual application template categories that were created in previous versions of DataQuant are contained in the Visual Application Templates folder under the Templates node. You can edit them, and then save them as visual dashboards to a repository or file. You cannot add new visual application templates in DataQuant.

The Visual Application Templates folder is not supported in the new repositories that you create.

Creating visual reports

Visual reports are page-based reports that include both text and graphics.

About this task

Using DataQuant for Workstation, you can quickly design visual reports using the intuitive Visual Designer perspective that includes an editor that presents both design and runtime views of the report; the Project Explorer view which details the structural content of each visual report; and the Palette, Properties, and Output views which support the Visual Designer editor and assist in creating visual reports.
Note: You can generate and view previously designed visual reports in DataQuant for WebSphere, but you cannot design visual reports. The Visual Designer perspective is not available.

The following steps outline the tasks that you will perform when creating or editing visual reports:

**Procedure**

1. Activate a visual report in the editor window. You can activate a visual report in the workstation editor window in one of the following ways:
   - Create a new visual report from a set of query results using the Display Report wizard.
   - Create a new visual report using the New Visual Report wizard.
   - Open an existing visual report.
   - Import a legacy visual report.

2. Open the Visual Designer perspective. The Visual Designer perspective presents an editor window, several views and the menus that you will use when you work with visual reports. You can open the Visual Designer perspective by using either the Open perspective toolbar button or by selecting Window > Open Perspective > Visual Designer.

3. View the key components of the visual report as listed in the Project Explorer view. Each visual report is listed under the Visual Reports node in the Project Explorer tree. Each visual report will have the following folders:
   - Connections
   - Queries
   - Globals
   - Main Page
   - Fixed Pages

4. Create data source connections. Visual reports are intended to be shared across the enterprise. In order to facilitate sharing and distributing visual reports, DataQuant for Workstation does not tie the queries that are included in a visual report to a specific data source. Instead it ties the query to a connection information alias. You must set up a connection information alias for each data source that will be used to obtain data for the visual report. All connection information aliases that are available for a report are listed in the Connections folder for the report in the Project Explorer view. You can set up a connection information alias for a data source using the Insert Connection dialog or by dragging the data source from the Workspaces view to the Connections folder for the visual report in the Project Explorer view.

5. Specify the main query that will supply the data for the visual report. You can specify an existing query or create a new query. When you specify the query you will be asked to associate the query with a connection information alias. The connection information alias will identify the data source against which the query will be run. The main query for a report is listed in the Queries folder for the visual report in the Project Explorer view. You can specify the query using the Insert Query dialog or by dragging the query from the Workspaces view to the Queries folder for the visual report in the Project Explorer view.

6. Design the main page of the report. The main query provides the data for the report and this data is formatted based on the design of the main page. See "Designing the main page of a visual report" on page 242 for more information on designing the main page.
7. Design one or more fixed pages that will be included in the report. Fixed pages are single pages with unique elements that can be inserted in the visual reports. See "Designing a fixed page for a visual report" on page 243 for more information on designing fixed pages.

8. Each time that you click the Preview button you are actually running the visual report object and you can view how your results will appear.

9. When you have completed designing or editing the visual report, you can save the report, print the report, or export the report to a HTML or PDF file.

Creating reports using the visual report wizard

The Create New Visual Report wizard will help you create a new visual report.

About this task

To create a new visual report using the Create New Visual Report wizard:

Procedure

1. Open the Create New Visual Report wizard in one of the following ways:
   - With an active query in the editor window, select Query > Transfer To > Visual Report.
   - Click the New Visual Report toolbar button.

2. Specify the name of the new visual report in the Visual report name field.

3. Select the template that will be used for the new visual report in one of the following ways:
   - Select the Predefined tab. From the list of predefined templates, select the one on which you want to model the new visual report. Predefined templates are delivered with the application. The layout of predefined templates cannot be changed. The Blank predefined template is the default template for all visual reports. It contains no structural elements.
   - Select the Local tab. From the list of local templates select the one on which you want to model the new visual report. Local templates are those that have been created for your organization and saved locally. Each template contains predefined content (such as company logos, standardized text, frequent graphical layouts). The templates are listed by category for organizational purposes. Users can edit the structural content of local templates to suit their needs.
   - Select the Repository tab. From the list of templates that are saved in the repository select the one on which you want to model the new visual report. Templates listed on the repositories page are those that have been created for your organization and saved in the repository. Each template contains predefined content (such as company logos, standardized text, frequent graphical layouts). The templates are listed by category for organizational purposes. Users can edit the structural content of repository templates to suit their needs.

Note: The Local tab is not displayed if there is no locally saved templates in your Templates directory.
4. Click Finish. A new visual report object opens in the editor. The design template for the main page is displayed. The Project Explorer view opens (if it is not already open) and lists the new visual report under the Visual Reports node in the tree.

Creating a new visual report template

Users can create visual reports with predefined content, such as company logos, standardized text, and special graphics. Each visual report can then be saved as a template that can be used by others to facilitate easy authoring of visual reports.

About this task

To create a new visual report template:

Procedure

1. Create a new visual report and include the common elements or open an existing visual report that will serve as the basis of the template.
2. Select File > Save As. The Save wizard opens.
3. Specify where you will save the visual report template. Select Save to Repository to save the report in a repository. Select Save to File to save the visual report locally.
4. Select the Save as template check box to save the visual report as a template.
5. Click Next. The second page of the wizard varies depending on where you are saving the visual report template. If you are saving the template locally, the Enter file name or choose file within a file dialog page of the wizard opens. You will specify the path to your local directory where you will save the template in the Path field. For example, in a Windows operating system the path would be:
   - For Windows XP or earlier:
     C:\Documents and Settings\[UserName]\Application Data\IBM\DataQuant for Workstation\Templates\VR Templates\VisualReport1
   - For Windows Vista or later:
     C:\Users\[UserName]\Application Data\IBM\DataQuant for Workstation\Templates\VR Templates\VisualReport1

   You can replace VisualReport1 with any name. It is recommended that the path be to the local directory as this is where DataQuant for Workstation will look for all local templates in order to display them to users when they create a new visual report. You can use the Browse button to search for a location.
6. If you are saving the template to a repository, the Set up the repository object parameters page of the wizard opens. The Select parent entry list displays a list of categories that you have set up to organize your repository templates. If you have not set up any categories, this list is blank. Click the New Visual Report Template Categories icon to create a category. Specify a name for the report in the Name field. Optionally you can specify a comment in the Comment field.
8. Specify a name for the template category. Optionally you can specify a comment in the Comment field.
9. Click Finish. The new visual report template category has been created. It is displayed in the Select parent entry list.
10. Specify a name for the report in the Name field. Optionally you can specify a comment in the Comment field.
11. Click Finish. The new visual report template is saved. It will be presented to users when they create a new visual report.

**Opening a legacy visual report**

Legacy visual reports are visual reports that were created using versions of QMF applications prior to 9.1.

**About this task**

You can open legacy reports directly or by importing the report. Once opened in DataQuant for Workstation V1.1 (or later), these reports can only be saved in a repository. They cannot be saved in a QMF catalog because once opened in V1.1 they cannot be opened in previous versions of QMF applications.

The following tasks describe opening legacy visual reports:

**Opening legacy visual reports directly**

You can open legacy reports directly.

**About this task**

To open a legacy visual report directly:

**Procedure**

1. Select File > Open From. From the pop-up menu select one of the following options depending on where the visual report object resides:
   - QMF catalog
   - Repository
   - File
2. One of the following three windows opens depending on your selection:
   - If you selected QMF catalog, the Open from QMF Catalog window opens. In the Data Source field specify the name of the data source where the QMF catalog resides. In the Owner field specify the owner of the object that you want to open. In the Name field specify the name of the object that you want to open. Select the visual report object. The visual report opens in the editor window. Select the Run Immediately check box to run the visual report when it is opened. The visual report opens in the editor window. If you selected the Run Immediately check box the result set is displayed.
   - If you selected Repository, the Open from Repository window opens. The contents of the current repository are displayed in a tree structure in the Select repository entry(s) list box. Expand the folders until you uncover the legacy visual report object that you want to open. Select the visual report object. The visual report opens in the editor window.
   - If you selected File, an operating system specific Open window opens. Type the full path name for the file in the file name field. You can use the browse function to search for the location of the file. Select the visual report object. The visual report opens in the editor window.
3. The legacy visual report opens. It can no longer be opened in a version of QMF prior to V9.1.
Importing legacy visual reports
You can import legacy visual reports.

About this task
To import a legacy visual report:

Procedure
1. From any perspective, select File > Import. The Import window opens.
4. In the File path field, specify the path to the visual report file that you want to import. You can use the Browse button to search for the file.
5. Type a unique name for your new visual report that will be created from the legacy visual report in the Report name field.
6. Type the location where you want to save the new visual report (within the currently connected repository) in the Location field.
7. Click Finish. A folder with the visual report name is added to the Visual Reports node in the Project Explorer view. For each new visual report, folders named Connections, Queries, Globals, and Scenes are also added in the Project Explorer tree. The editor window opens with a canvas where you can make edits to the converted report.

Specifying data source connections for visual projects
You set up a connection information alias for each data source that will provide data for the visual project (visual report or dashboard).

About this task
When you add the queries that will be used in a visual project, you must specify what connection information alias will be associated with each query.

To change the data source that will be used for a query, you edit the query information and select a connection information alias entry that points to the data source that you want to use. This feature allows users to easily share visual reports and use their own data sources. In addition, it facilitates moving between test and production data sources.

To specify a connection information alias for a data source, perform the following steps:

Procedure
1. Open the Insert Connection window.
   You can open the Insert Connection window in one of the following ways:
   • From the Project Explorer view, right-click the Connections node for the specific visual project. Select Insert Connection from the pop-up menu.
   • From the Project Explorer view, double-click the Connections node. The Insert Connection window opens.
   • Click a query from the Workspaces or Repository Explorer view. With the mouse button pressed, drag the query to the Queries node in the Project Explorer view. A new data source connection is automatically added to the Connections folder of the Project Explorer view.
2. From the list of available data sources, select the data source that will be associated with this connection information alias.

3. Specify a unique name for this connection information alias in the Connection name field.

4. Click Finish. The Project Explorer view lists the new connection information alias under the Connections node in the tree.

Note: You can also set up a connection information alias for a data source by dragging the data source from the Workspaces view to the Connections folder for the visual project in the Project Explorer view. The connection information alias is added with the name ConnectionN. You can edit the name of the connection information alias by right-clicking the entry and selecting Rename.

Results

You have set up the data source connection information for the visual data object.

Specifying queries for visual projects

Visual projects (visual reports or dashboards) can include one or more queries to run to obtain the result data and display it in the visual project.

About this task

You can create a new query that will be included in the visual project or you can use existing queries.

To specify a query for a visual project, perform the following steps:

Procedure

1. From the Project Explorer view, right-click the Queries node and select Insert Query from the menu.

2. Specify a unique name for the query in the Query name field.

3. Identify the query and specify where it resides in one of the following ways:
   a. If you are creating a new query, select Create a new query.
   b. If you are creating a new visual query, select Create a new visual query.
   c. If you are creating a new analytical query, click Create a new analytical query.
   d. If you are creating a new Cloudant query, click Create a new cloudant query.
   e. If you are creating a new forecast, click Create a new forecast.
   f. If the query exists, select Attach to an existing query. The Select query list appears. Specify the name of the query and where it resides in the Select Query field. You can click Browse to search for the query.
   g. If you would like to insert a static query, select From file.
      - Select Insert Data to copy the query directly into the Queries folder. This option provides greater portability, because the query will stay with the visual project if it is moved from one system to another.
      - Select Use link to file to create a link to the query’s system directory location. This option limits portability, because if the visual project is moved from one system to another, the query will no longer be available.
      - Specify the query to be added in the Select data file list. You can click Browse to search for the query.
**Note:** You can open DBF, IXF, XML, and TAB files and use them as queries. You can edit the imported result set by filtering, adding calculated columns, applying grouping and aggregation. When you save the visual project, these files can be converted to dynamarts and saved with the visual project. All modifications will be preserved.

4. If you insert a visual query or Cloudant query, you must select a data source connection that you want to use for the query from the **Connection** list. If the **Connection** list is empty, add a data source connection to a visual project.

5. Click **Add Connection** to open the Insert Connection window.

6. From the data sources tree, select the data source that you want to add for the visual project and click **Finish**. The data source connection that you selected is added to the visual project and displayed in the **Connection** list.

7. If you are inserting an existing forecast or analytical query into a visual project, you can edit connection settings for subordinate queries in the Edit Connections window.

8. If you are inserting a new forecast or analytical query, you cannot set connections for subordinate queries because these queries are not specified yet.

9. Click **Finish**. The Project Explorer view lists the query under the **Queries** node in the tree. If the query requires a parameter, a **Parameters** folder is added. You will automatically save the query when you save the visual project.

**Results**

**Note:** You can also add a query by dragging the query from the Workspaces view to the **Queries** node of the visual project in the view. The selected query is added to the **Queries** folder. A new data source connection is automatically added to the **Connections** folder of the Project Explorer view.

### Designing the main page of a visual report

The main page of a visual report is where you specify the content for the overall report. You will specify the header, detail, and footer template information that will be repeated for each page of the report.

**About this task**

A visual report main page is divided into the following sections:

- **Report Header** section describes what appears at the beginning of the report.
- **Page Header** section describes what appears at the top of each page of the report.
- **Detail Header** section describes what appears as column headers to each detail line.
- **Detail** section describes how each detail line of data will be presented.
- **Detail Footer** section describes what appears on the break line for a detail section.
- **Page Footer** section describes what appears at the bottom of each page of the report.
- **Report Footer** section describes what appears at the end of the report.

The process of designing each section of the main page of a visual report is the same for each section. To specify what should appear in each section of the visual report:
Procedure

1. You can select which section you will be designing in one of the following ways:
   - From the Project Explorer tree, select the folder for the section of the report that you want to design.
   - In Design mode, the editor window is divided into multiple sections each labeled and associated with a section of the report. Click in the section that you want to design.

2. In each section of the report, you can insert design elements that identify what will be included in that section of the report. You choose the design elements from the Palette view. Select the objects that you want to insert. Then click in the section of the report where you want to place the object.

3. The properties of each section of the visual report and the properties of each object that you can insert in the report are displayed in the Properties view. You can modify the properties of each section of the visual report and the properties of each object in the Properties view. The properties of some of the report sections and objects can also be changed in the editor window.

4. You can view your report at any time during the design phase. Click the Preview tab in the editor window to display the report in preview mode. To return to the design mode, select the Design tab.

5. You can save the report to a file, in the QMF catalog if it is available, or in a repository. You can also print the report or save it as an HTML or a PDF file.

Designing a fixed page for a visual report

Fixed pages of a visual report are single pages with unique elements that can be inserted in the visual report. Fixed pages can be used for example as cover pages, chapter beginning pages, special graphic pages, or a report closing page.

About this task

A visual report fixed page is divided into the following sections:
   - PageHeading section describes what appears at the top of the fixed page.
   - Fixed1 section describes what appears in the first fixed section of the page.
   - Fixed2 section describes what appears in the second fixed section of the page.
   - PageFooting section describes what appears at the bottom of the fixed page.

The process of designing each section of a fixed page is the same for each section. You can create as many fixed pages as your report requires. You specify where the pages will appear in the report using the property FixedPages that is available for each of the main page sections.

To create a fixed page for your visual report:

Procedure

1. When you create a new visual report object, you automatically get a folder in the Project Explorer that can be used to design your first fixed page. You can however create as many fixed pages as is required by your report. To create additional fixed pages, right-click the Fixed Pages node from the Project Explorer view. Select Insert Fixed Page from the pop-up menu. The New Fixed Page wizard opens. Specify a unique name for this fixed page in the Fixed page name field. Click Finish. The Project Explorer view lists the new fixed page under the Fixed Pages node in the tree.
2. Select which section you will be designing in one of the following ways:
   - From the Project Explorer tree, click the folder for the section of the fixed page that you want to design.
   - In the Design mode, the editor window is divided into multiple sections each labeled and associated with a section of the fixed page. Click the section that you want to design.

3. In each section of the fixed page, you can insert design elements that identify what will be included in that section of the page. You choose the design elements from the Palette view. Select the objects that you want to insert. Then click the section of the fixed page where you want to place the object.

4. The properties of each section of the fixed page and the properties of each object that you can insert in the report are displayed in the Properties view. You can modify the properties of each section of the fixed page and the properties of each object in the Properties view. The properties of some of the fixed page sections and objects can also be changed in the editor window.

5. Specify where to insert the fixed page in the main report using the property FixedPages that is available for each of the main page sections. If you specify a FixedPages property in the:
   - Report header section: the fixed page will appear as the cover page of the report.
   - Page header section: the fixed page will appear before the highest level break.
   - Detail header section: the fixed page will appear before the detail level break.
   - Detail footer section: the fixed page will appear before the detail level break.
   - Page footer section: the fixed page will appear after the highest level break.
   - Report footer section: the fixed page will appear as the last page of the report.

6. You can view your fixed page at any time during the design phase. Click the Preview tab in the editor window to display the report in preview mode. To return to the Canvas mode, select the Design tab.

Managing report data retrieval

You can control when data is retrieved for visual reports.

About this task

By default, visual reports are run against live data. Each time the visual report is run, queries are run, and the data in the report is updated. You control the data retrieval for a visual report using a set of expiration schedules. This option allows you to specify when, if at all, you want to run the visual report queries to update the data that is displayed in the report.

To set up expiration schedules for a visual report's data retrieval:

Procedure

1. Create a new visual report, including the queries that will be used to retrieve data.
2. From the Project Explorer view, expand the Globals folder, and right-click the Expiration Schedules node for the visual report. Select Insert Schedule from the context menu. The Expiration schedule configuration wizard opens.
3. You can create one or more schedules that will control the data retrieval for the visual report. Specify the name of the schedule in the Schedule name field. This can be any name that has meaning.

4. The type of schedule controls when data that has been retrieved for this visual report will expire. You can choose one of the following schedule types by selecting one of the Data expires radio buttons:

   - **daily**: Select this option to specify that the same data will be used for a 24 hour period. If you select this option, the visual report will run, data will be retrieved, and every time the report is run in the next 24 hour period it will use the same data. When the 24 hour period expires, the next time the report is run, new data will be retrieved.

   - **weekly**: Select this option to specify that the same data will be used for a period of 7 days. If you select this option, the visual report will run, data will be retrieved, and every time the report is run in the next 7 days it will use the same data. When the 7 day period expires, the next time the report is run, new data will be retrieved.

   - **monthly**: Select this option to specify that the same data will be used for a month. If you select this option, the visual report will run, data will be retrieved, and every time the report is run in the next month it will use the same data. When the month expires, the next time the report is run, new data will be retrieved.

   - **at specific date**: Select this option to specify that the same data will be used until a specific date is reached. If you select this option, the visual report will run, data will be retrieved, and every time the report is run up until the specific date it will use the same data. When the specific date is reached, the next time the report is run, new data will be retrieved.

   - **never expires**: Select this option to specify that the same data will always be used. If you select this option, the visual report will run, data will be retrieved, and every time the report is run it will use the same data.

   - **always expires**: Select this option to specify that the data will always be updated every time the report is run.

   - **after the visual project is closed** to have the schedule expire after you close the visual project.

5. If you selected **daily**, **weekly**, **monthly**, or **at specific date**, click the Next.

6. Depending on the type of schedule that you are creating one of the following variations of the Set schedule properties page of the wizard opens:

   - If you selected **daily**, you will specify the date that the 24 hour period will start in the Begin date field and the time the 24 period will start in the Begin time field. You specify when the 24 hour period will occur using the Data expires radio buttons. Select daily to specify that the 24 period will occur every day. Select each to specify that the 24 hour period will occur every number of days. Select the number from the drop-down list. Select days of week to specify that the 24 hour period will occur on one or more specific days. You must select the check boxes of the days.

   - If you selected **weekly**, you will specify the date that the week will start in the Begin date field and the time the week will start in the Begin time field. You can select a number from the each week list to specify that the 7 day period will occur every number of weeks. You can select a day from the days of week check boxes to specify that your 7 day period will begin on a specific day of the week.

   - If you selected **monthly**, you will specify the date that the month will start in the Begin date field and the time the month will start in the Begin time field.
field. You can select or to specify that data retrieval will occur on a specific day of the week in the month. Select the month from the select months list of check boxes.

- If you selected at specific date, you will specify the date and time that will trigger data retrieval in the Date and time list box. Click the cell to bring up a calendar where you will select the date and time.

7. Click Finish. The schedule is created and listed in the Project Explorer view. You will repeat this process to define additional properties. After all schedules are defined you will assign the schedules to the queries in the visual report.

Assigning data retrieval schedules to visual report queries
To control visual report data retrieval, you must assign the data retrieval schedules to the queries in the visual report.

About this task
To assign data retrieval schedules to visual report queries:

Procedure
1. Right-click the Queries node for the visual report in the Project Explorer view. Select Assign Expiration Schedules from the pop-up menu. The Manage expiration schedules wizard opens.
2. Select Use expiration schedules to assign an expiration schedule to the visual report.
3. Select the default expiration schedule from the Default expiration schedule list. This schedule will be used to control the data retrieval for all queries that are included in the visual report unless they have been assigned a specific schedule.
4. You can assign specific schedules to specific queries. Each query that is included in the visual report is listed in the Query name cell. To assign a specific schedule to the query, click in the corresponding Schedule name cell. Select the unique schedule from the drop-down list of available schedules.
5. Click Refresh cache to refresh the retrieved data for the query.
6. Click Finish. Any schedule assignments that you have made are saved.

Converting visual reports to offline reports
You can convert a visual report to an offline report. An offline report is a static version of a visual report. Users can open the report and will always see the same data.

About this task
To create an offline report from a visual report:

Procedure
1. Open a visual report.
2. Select Edit > Convert to Offline Report. All the queries for the visual report will be run and all the data that is needed to render the report will be retrieved.
3. Save the offline report in the repository. You can not save the report in a file.
4. To convert an offline report back to an online report, open the offline report and select Edit > Convert to Online Report.
Working with Ad hoc reports

Ad hoc reports are canvas-based reports that graphically represent information retrieved from a database. An Ad hoc report includes a scene with a canvas.

Ad hoc reports provide an extempore and user-friendly way to design reports. You can create Ad hoc reports quickly by using the intuitive runtime editor designed to make working with data objects more efficient.

When creating a report, you can place multiple objects on the Canvas and manage the Canvas space to suit to your purposes. Typically, the Canvas represents what you believe to be the most appropriate set of data objects.

The Canvas Internal Tree displays the contents of your home workspace and allows you to add the objects contained in it to your report.

The Canvas provides the following objects:
- Queries
- Visual Queries
- Analytical Queries
- Dynamarts
- Compound Dynamarts
- Drill-down Paths
- Tables

The Canvas with all the objects and layouts is a part of the Ad hoc report solution and is saved as a Built-in Canvas type directly to that report.

You can also wire objects in the Canvas for Ad hoc reports.

Creating Ad hoc reports

You can create an Ad hoc report and populate it with objects in the Ad hoc Reports runtime editor.

About this task

An Ad hoc report includes a scene with a canvas. When creating an Ad hoc report, you can place multiple objects on the Canvas and manage the Canvas space according to your purposes.

To create an Ad hoc report perform the following procedure:

Procedure

1. Open the Ad hoc Report editor in one of the following ways:
   - Select File > New > Other.
     The New window opens.
     Expand DataQuant Objects and select Ad hoc Report
     Click Next.
   - Click the New Ad hoc Report toolbar button ( ).
     The Ad hoc report opens in the editor.
2. Drag your cursor on to the Canvas object. The Canvas changes color from white to gray and a cross appears in the center of the Canvas indicating the position of the first display.

3. Left-click anywhere in the grey display area. The Canvas’s Internal Tree is displayed.

4. Expand the folders in the Internal Tree to locate the object that you want to add to the Canvas. Canvas supports the following objects:
   - Queries
   - Visual Queries
   - Analytical Queries
   - Dynamarts
   - Compound Dynamarts
   - Drill-down Paths
   - Tables

5. Double-click the object that you want to insert into the Ad hoc report. The default Display Mode for the object you selected is rendered graphically.

6. Click any of the widget icons listed on an object’s Display Mode and the image is rendered accordingly on the Canvas.

   **Note:** The widget icons that are displayed on the toolbar vary depending on the type of object you added to the Canvas. The complete set of widgets allows the following actions:
   - Switching display mode to one of available display modes
   - Changing chart type to one of available 2d charts
   - Changing chart type to one of available 2d with depth charts
   - Changing the query columns and aggregation being charted
   - Deleting the query result from the canvas
   - Adding a new chart
   - Navigating forward and backward in Drill-down Paths.

**Results**

You have created an Ad hoc report.

**Related concepts:**

[“Managing Canvas operations using the Canvas toolbar” on page 348](#)

The Canvas toolbar provides actions for managing the Canvas in ad hoc reports and visual dashboards.

**Related tasks:**

[“Wiring objects on the Canvas” on page 350](#)

Wiring objects defines the relationship and dependencies between objects. Use the object Wiring Mode to connect objects on the Canvas.

**Working with visual dashboards**

A Visual dashboard is a type of visual project that presents interactive or persistent data to a wide range of users.
About this task

Visible dashboards can simultaneously draw data from multiple, heterogeneous data sources and present the data using a variety of graphs, charts, and graphical widgets. Using DataQuant for Workstation, content developers can create a visual dashboard that can be viewed by multiple users with either DataQuant for Workstation or DataQuant for WebSphere.

**Note:** You can view previously designed visual dashboards in DataQuant for WebSphere, but you can not design them. The **Visual Designer** perspective is not available.

The following steps outline the tasks that you will perform when creating or editing visual dashboards:

**Procedure**

1. Activate a visual dashboard in the workstation editor window. You can activate a visual dashboard in the workstation editor window in one of the following ways:
   - Create a new visual dashboard using the Create New Visual Dashboard wizard.
   - Open an existing visual dashboard.

2. Open the **Visual Designer** perspective. The **Visual Designer** perspective presents an editor window and several views and menus that you will use when you work with visual dashboards. You can open the **Visual Designer** perspective by using either the **Open perspective** toolbar button or by selecting **Window > Open Perspective > Visual Designer**.

3. View the key components of the visual dashboard as listed in the **Project Explorer** view. Each visual dashboard is listed under the **Visual Dashboards** node in the **Project Explorer** tree. Each visual dashboard will have the following folders:
   - **Connections**
   - **Globals**
   - **Queries**
   - **Scenes**

4. Create data source connections.

Visible dashboards are intended to be shared across the enterprise. In order to facilitate sharing and distributing visual dashboards, DataQuant for Workstation does not tie the queries that are included in a visual dashboard to a specific data source. Instead, it ties the query to a connection information alias.

You must set up a connection information alias for each data source used to obtain data for the visual dashboard. For information about setting up a connection information alias, see **Specifying data source connections for visual projects**

5. Specify the first query that will supply data for the visual dashboard.

You can specify an existing query or create a new query. When you specify the query the system prompts you to associate the query with a connection information alias. The connection information alias identifies the data source against which the query runs. For information about setting up queries, see “Specifying queries for visual projects” on page 241.
6. Design the first scene for the dashboard. A visual dashboard consists of one or more scenes.

A scene is the container (similar to a presentation slide) that will hold all the elements used to display your data. After creating a new visual dashboard, the system provides one default scene. You can choose to have only one scene in your visual dashboard or you can add additional scenes.

7. Insert objects into a scene.

The objects that you can insert are displayed in the Palette view and are organized on individual palettes based on their type. From the Palette view select the objects that you want to insert into your scenes.

8. Modify the object's properties in the Properties view.

By modifying an object's properties you set the values that determine how the object looks and behaves. You can use the Expression Designer to help you specify an object's properties.

9. Insert the layout objects

In a visual dashboard scene layout objects display query results.

When you insert a layout object the system prompts you to associate the object with a query that is contained in the Queries folder. You can add queries to your Queries folder at any time during the process of creating your dashboard.

Layout objects can present data in many ways. For most layout objects, you display the results of multiple queries in a single layout. For example, you might create a single XY chart that displays sales figures derived from one query and spending figures derived from another query. You can add multiple layout objects. When you place layout objects, you can pass query result information from a higher level layout object to a lower level layout object. Because you have the ability to pass this information, you can use the placed layout objects to display more detailed information that relates to a specific data value.

10. Use the Events view to assign the different navigation options.

By assigning navigation options to scenes and objects, you set a path through the dashboard. One form of navigation is to define dependencies between dashboard control objects (such as combo or listbox objects) and data-driven layout objects (such as a grid or bar chart object) using the Connectivity tool from the Palette view.

11. Run the visual dashboard.

Each time that you click the Runtime tab, you are actually running the visual dashboard object. By performing this step you can review how the results will display at run time. Think of this step as a test-run of your dashboard that allows you to improve upon the design before saving and deploying the dashboard.

12. Save the visual dashboard.

When you have completed designing or editing the visual dashboard, you can save the dashboard and copy it to one or more workspaces for distribution to other users.

Planning visual dashboards

Before developing a visual dashboard, plan your development and design process. One of the most important planning activities for designing visual dashboards is known as storyboarding.
Using storyboards

Storyboards are visual aids to a design and a design process. Acting as a graphical organizer, storyboards help all members of your team pre-visualize the dashboard that you intend to design.

The information in this section is intended as a general guideline to using storyboards.

Proper use of storyboards during the planning phase facilitates the design process by clearly illustrating how the dashboard will look and function.

Storyboards help take the guesswork out of the actual design phase and can minimize potential development missteps and help you deploy the dashboard more efficiently. Additionally, a well-designed dashboard has a higher likelihood for reuse in other areas of the enterprise.

You use a storyboard to design the basic features, functionality and presentation flow of the scenes and objects to be included in your visual dashboard.

A storyboard for a visual dashboard should specify the following:

- Data for each scene
- Scenes, drawn roughly
- Layouts for each scene
- Sequence of presentation
- Points where users can access further information
- Actions that result from user events
- Jumps between scenes

There are many types of tools, with varying level of complexity, that you can use to create storyboards. You can use a pencil and paper to sketch the design and flow. You can use a presentation tool, such as Microsoft PowerPoint, to create sample layouts, with supporting detail on notes pages. You can also put together Web pages to show actual jumps and navigation flow.

Use the following list of general topics as a guide to elements you should include in your storyboard / design:

**Displaying query results data**

You use layout objects or the List and Combo controls to display query results data. Explore the different display options and decide which ones best suit the kind of analysis you want to show.

**Capturing user input**

You can capture user preferences with standard user interface controls. Look at the Controls palette on the Palette view to explore your options.

**Passing information**

Using parameters, you can pass information acquired from user actions to affect the display of a dashboard, the contents of a scene, or the execution of a query.

**Navigating around dashboards**

You can use navigation features to allow users to move around dashboards to get to new information.
Locating your data

Your storyboard should specify the database and include a list of the tables that you plan to use. Writing this list will help you organize your work. You can then spend a session creating all the queries you need prior to creating your first scene. Alternatively, you might work with others who extract the data that you need from the database tables and consolidate it in summary tables.

Creating Visual Dashboards using the Visual Dashboard wizard

You can create a new Visual Dashboard using the Create New Visual Dashboard wizard.

About this task

To create a new Visual Dashboard, perform the following steps:

Procedure

1. Open the Create New Visual Dashboard wizard on one of the following ways:
   - Click the New Visual Dashboard toolbar button ( )
2. Type a unique name for your dashboard in the Dashboard name field.
3. Select the template that will be used for the new visual dashboard in one of the following ways:
   - Select the Predefined tab. From the list of predefined templates select the one on which you want to model the new visual dashboard. Predefined templates are delivered with the application. The layout of predefined templates cannot be changed. The Blank predefined template is the default template for all visual dashboards. It contains no structural elements.
   - Select the Local tab. From the list of local templates select the one on which you want to model the new visual dashboard. Local templates are those that have been created for your organization and saved locally. Each template contains predefined content (such as company logos, standardized text, frequent graphical layouts). The templates are listed by category for organizational purposes. Users can edit the structural content of local templates to suit their needs.
   - Select Repository tab. From the list of templates that are saved in the repository select the one on which you want to model the new visual dashboard. Templates listed on the repositories page are those that have been created for your organization and saved in the repository. Each template contains predefined content (such as company logos, standardized text,
frequent graphical layouts). The templates are listed by category for organizational purposes. Users can edit the structural content of repository templates to suit their needs.

4. Click Finish. A folder with the dashboard name is added to the Dashboards node in the Project Explorer view.

For each new dashboard a folder is created for the Connections, Queries, and Globals. In addition, a Scenes folder is added with a default first scene. The editor window opens with a canvas where you will design the first scene of the dashboard.

Results

You have created a new Visual Dashboard using the Create New Visual Dashboard wizard.

What to do next

You are now ready to design your dashboard, adding Connections, Queries and Globals and additional Scenes.

Creating a new visual dashboard template

Users can create visual dashboards with predefined content, such as company logos, standardized text, and special graphics. Each visual dashboard can then be saved as a template that can be used by others to facilitate easy authoring of visual dashboards.

About this task

To create a new visual dashboard template:

Procedure

1. Select File > Save As. The Select a wizard window opens.

From the Select a wizard window, select which wizard to use, select the Save as template check box and then click Next:

   • Save to File
   
   Select this option if you want to use the Save to File wizard to save the Visual Dashboard locally to a file.
   
   See step 2

   • Save to Repository
   
   Select this option if you want to use the Save to Repository wizard to save the Visual Dashboard to a repository.
   
   See step 3 on page 254

2. On the Save to File page, enter the file location in the Path: field or select Browse. to locate the file and click Finish.

For example, in a Windows operating system the path would be one of the following:

   • For Windows XP or earlier:
     C:\Documents and Settings\[user_name]\Application Data\IBM\DataQuant for Workstation\Templates\VD Templates\VisualDashboard1

   • For Windows Vista or later:
     C:\Users\[user_name]\Application Data\IBM\DataQuant for Workstation\Templates\VD Templates\VisualDashboard1
You can replace VisualDashboard1 with any name. It is recommended that the path be to the local directory as this is where DataQuant for Workstation will look for all local templates in order to display them to users when they create a new Visual Dashboard. You have saved the Visual Dashboard template to a file.

3. On the Save to Repository page, set up the repository object parameters.
   The Select parent entry list displays a list of categories that you have set up to organize your repository templates.
   If you have not set up any categories this list is blank.

4. Conditional: Click the Create New Visual Dashboard Template Categories icon to create a category.
   The Create New Visual Dashboard Template Category wizard opens.

5. Specify a name for the template category. Optionally you can set a comment in the Comment field.

6. Click Finish.
   The new Visual Dashboard template category has been created. It is displayed in the Select parent entry list.

7. Specify a name for the dashboard in the Name field. Optionally you can set a comment in the Comment field.

8. Click Finish.
   The new Visual Dashboard template is saved. It will be presented to users when they create a new visual dashboard.

**Importing a legacy Visionary world**

You can create a new dashboard from an existing Visionary world that was created using a previous version of QMF Visionary.

**About this task**

The Visionary world that you import must have been exported as an XML file using QMF for Visionary Version 8.1, Fix Pack 13 or later. To export a Visionary world using QMF for Visionary Version 8.1, Fix Pack 13, you select File > Export as XML.

To import a legacy Visionary world:

**Procedure**

1. From any perspective, select File > Import. The Import window opens.
2. Select Visionary World File.
3. Click Next. The Visionary World Import wizard opens.
4. In the File path field, specify the path to the Visionary world that you want to import. You can use the browse button to search for the file.
5. Type a unique name for the new dashboard that will be created from the legacy Visionary world in the Dashboard name field.
6. Type the location where you want to save the new dashboard (within the currently connected to repository) in the Location field.
7. Click Finish. A folder with the dashboard name is added to the Dashboards node in the Project Explorer view. For each new dashboard, folders named Connections, Queries, Globals, and Scenes are added. The editor window opens with a canvas where you can make edits to any scene within the converted world.
Specifying data source connections for visual dashboards

You set up the connection information alias for each data source that will be used in the visual dashboard.

About this task

Visual dashboards are intended to be shared across the enterprise. In order to facilitate sharing and distributing visual dashboards, DataQuant for Workstation does not tie the queries that are included in a visual dashboard to a specific data source. Instead, it ties the query to a connection information alias. The connection information is saved with the visual dashboard and can be easily modified to point to any data source.

The connection information includes a name for the connection and the name of the data source to which it will point. When you add the queries that will be used in the visual dashboard, you will specify what connection information alias will be associated with the query.

To change the data source that will be used for a query, you edit the connection information and point to the data source that you want to use. This feature allows users to easily share visual dashboards and use their own data sources. In addition, it facilitates moving between test and production data sources.

To specify connection information for a data source:

Procedure

1. You can open the Insert Connection window in one of the following ways:
   - From the Project Explorer view, right-click the Connections node. Select Insert Connection from the pop-up menu. The Insert Connection window opens.
   - From the Project Explorer view, double-click the Connections node. The Insert Connection window opens.
2. From the list of available data sources, select the data source that will be associated with this connection information alias.
3. Specify a unique name for this connection information in the Connection name field.
4. Click Finish. The Project Explorer view lists the new connection information alias under the Connections node in the tree.

Note: You can also set up a connection information alias for a data source by using one of the following methods:
   - Drag a data source from the Workspaces view to the Connections folder of the visual project in the Project Explorer view.
   - Drag a query from the Workspaces or Repository Explorer view to the Queries node of the visual project in the Project Explorer view.
      A new data source connection is automatically added to the Connections folder of the Project Explorer view.

Results

Note: The connection information alias is added with the name ConnectionN. You can edit the name of the connection information alias by right-clicking on the entry and selecting Rename.
Working with scenes

A visual dashboard consists of one or more scenes that will display data to users.

About this task

Upon creating a new visual dashboard, you are given one default scene. You can choose to have only one scene in your visual dashboard or you can add additional scenes. Within each scene of your dashboard you can add objects to display data and information to visual dashboard users and you can set up the navigation necessary for users to move from one scene or object to another.

This topic describes the following operations:

- Adding a scene to a visual dashboard
- Renaming a scene in a visual dashboard
- Deleting a scene from a visual dashboard

The following steps outline the tasks that you can complete when working with scenes:

Procedure

1. To add a scene to a dashboard, from the Project Explorer view, use the New Scene wizard.

    You can open the New Scene wizard in one of the following ways:
    - Right-click the Scenes folder for the visual dashboard to which you want to add the scene and select New Scene from the pop-up menu.
    - Double-click the Scenes folder for the visual dashboard to which you want to add the scene.

    The New Scene wizard opens.
    a. Specify a unique name for this scene in the Scene Name field.
    b. Click Finish.

    The Project Explorer view lists the new scene in the visual dashboard project. A new scene canvas opens in the editor.

2. To rename a scene, perform the following steps:
    a. Select the scene that you want to rename using one of the following methods:
       - From the Project Explorer view, right-click the scene that you want to rename and select Rename from the pop-up menu.
       - With an active scene in the editor window, select Edit > Scene > Rename
    b. Type the new name for the scene in the highlighted edit box. The scene is renamed and the new name is displayed.
    c. Click outside the edit box in the Project Explorer view.

    The scene is renamed and the new name displays.

    **Note:** Any references to the scene by its original name are not updated.

3. To delete a scene, complete either of the following steps:

    **Note:** You cannot delete the scene if it is the last scene remaining.
    a. From the Project Explorer view, right-click the scene that you want to delete and select Delete from the pop-up menu.
    b. With an active scene in the editor window, select Edit > Scene > Delete.
The scene is deleted from the visual dashboard project.

**Note:** Any references in the dashboard to the deleted scene are not updated.

**Using scene parameters**

Using parameters, you can alter the value of an object's properties in the scene. Scene parameters are available for use by only the scene in which it is defined.

**About this task**

A scene parameter is available for use by only the scene in which it is defined. Each scene that you create has a **Locals** folder that is added to the **Project Explorer** and includes all the parameters that are defined for the scene.

To create a scene parameter:

**Procedure**

1. Open the Insert Parameter window in one of the following ways:
   - From the **Project Explorer** view, right-click the **Locals** node for the scene to which you want to add a parameter. Select **New**. Select **Parameter** from the pop-up menu. The Insert Parameter window opens.
   - From the **Project Explorer** view, right-click the scene for which you want to create the parameter. Select **New**. Select **Parameter** from the context menu. The Insert Parameter window opens.
   - From the **Project Explorer** view, double-click the Parameters node for the scene to which you want to add a parameter. The Insert Parameter window opens.

2. Type a unique name for the scene parameter in the **Name** field.

3. Select a data type from the **Data type** list box. The data type will describe what type of data will be included in the parameter. You have the following options:
   - **Boolean**: True or False
   - **Color**: Color
   - **DateTime**: Date and time value
   - **DateTimeSet**: Multiple data and time values
   - **FilePath**: Path to a file
   - **FontName**: Valid font type
   - **Integer**: Whole number
   - **Literal**: As is data. This data type only applies to query parameters
   - **Number**: Floating point number
   - **NumberSet**: Multiple floating point numbers
   - **Percentage**: Value that will be used as a percentage
   - **Point**: Single x,y point value
   - **PointSet**: Multiple x,y point values
   - **PolySet**: Multiple point variables that can be used to construct a polygon
   - **Text**: A text string
   - **TextSet**: Multiple text strings

   The default data type is **Integer**.

4. If you want to add a description, enter the text in the **Description** field.
5. If you want to specify a default value, select the **Has default value** check box. A field is displayed where you can type a default value for the parameter. This value will be used if no other value is specified.

6. Click **Finish**.
   The Insert Parameter window closes. The specified parameter is added to the parameters node in the **Project Explorer** view.

**Creating scene views**

In visual dashboards you can create scene views. Use them to arrange and modify scene objects to adjust the view of the scene to different screen resolutions.

**Before you begin**

You create a scene on the **Canvas** tab and define the behavior of the scene objects. This view is the default view for the scene.

**About this task**

You can use scene views to customize the view of a visual project, and you cannot use them to change events and event actions that are defined in your visual project.

For each scene in a visual project you can create any number of required scene views.

To create a scene view:

**Procedure**

1. Open a scene for which you want to create a scene view in **Design** mode. The default view is displayed on the **Default View** tab.
2. Click the plus sign to open the Create Scene View wizard.
3. From the **Category** list, select the type of screen, and from the **Templates** list, select the screen resolution for which you want to create a scene view. Then go to step 9.
   You can create the following types of scene views:
   - Desktop
   - Tablet
   - Phone
   - Custom

   **Tip**: To display a scene correctly in web browsers of different mobile devices, you do not need to create scene views for all possible screen resolutions. You can select one template, for example, **iPhone 5, 5C, 5S** from the **Phone** category and create the landscape and portrait scene views for it. At runtime, the appropriate scene view is selected and scaled to screen resolutions of different mobile devices. To enable scene views scaling, ensure that the **ScaleToWindow** scene property is set to true.

   If you create a scene view for a scene that contains the embedded scene object, you must specify the exact size of an embedded scene in pixels for desktop, tablet, and phone presentations.
4. To add a specific scene view template, select **Custom** from the **Category** list and click **Add Custom View**.
5. Enter the template description in the View description field.
6. In the Width and Height fields, specify the width and height in pixels for your template.
7. Click Create View to add it to the Templates list, and then click Done.
8. Select the custom view that you created from the Templates list.
9. Click Finish to close the Create Scene View wizard and open the scene view tab. The scene view tab displays a canvas with a frame that defines the screen area.
10. On the scene view tab, modify the content of the scene to adjust it to the defined screen area. You can do the following:
   • Display or hide the scene objects for the view. To do this, select or clear the corresponding check boxes in the Visibility view.
   • Rearrange the position of the scene objects.
   • Modify scene objects properties.
11. When you completed creating the scene view, save it with your visual project.
12. To remove a scene view from your visual project, close the appropriate scene view tab.

Scene views selection for different devices:

The topic contains useful information about how the appropriate scene view is selected when you run a visual dashboard in a web browser of a mobile device or user machine.

When you run a visual dashboard:
1. DataQuant receives information about the screen size of a web browser that is launched on a mobile device or user machine.
2. From all scene views that are created for the appropriate scene, the scene views with the orientation that corresponds to the orientation of a web browser are selected.
3. From the list of scene views with the appropriate orientation, the scene view with the size that is close to the size of a web browser is selected.
4. The selected scene view is proportionally scaled by width to the size of a web browser.

Note:
• Scaling is available only in web browsers of mobile devices and only if you use web links to open visual dashboards.
• The default scene view is not scaled because scaling requires the exact size of a scene view, but the exact size of the default scene view is not defined until it is built in runtime.
• The scene views with too small size are not scaled properly. For different mobile devices, the minimum size of a scene view is different. It is recommended to check the scene view in a web browser of a mobile device to ensure its correct displaying and scaling.

Adding navigation features to your visual data objects
You design how the user will move through the dashboard by assigning navigation to scenes and objects.
About this task

Depending on how you want to present your data to your user, you can choose to apply any of the following navigation features to your scenes and objects:

Using the Drilldown wizard:

Although it can be done manually, the Drilldown wizard automates the process of creating the click event, identifying the destination scene, and mapping the required scene parameters.

About this task

The ability to drill down to additional information is a very important navigation feature. It enables users to click an object and jump to another scene that contains more information that relates specifically to the original object’s value. The Drilldown wizard is available for all layout objects that can display query result data (support data templates).

Note: You can also create click events, identify the destination scene and map the required scene parameters manually.

To set up a click event to a new destination scene using the Drilldown wizard:

Procedure

1. From the Project Explorer view, expand the layout object’s folder to which you want to add the click event navigation feature. Expand the data template folder for that layout object. Expand the level folder. Right-click the object to which you want to apply the click event and select Drilldown from the pop-up menu. You can also open the object in the editor and right-click the object in the editor window. The Drilldown wizard opens.

2. The first page of the wizard summarizes the steps necessary to create the click event.

3. Click Next.

   On the first page of the Drilldown wizard, select the query result data columns whose values you would like to pass through to your target scene. Any data values that you pass can be used by the target scene to deliver additional information that applies specifically to that originating data value. Use the arrow keys to move the data columns from the Available fields list box to the Fields to be displayed list box. Each data column that you add to the Fields to be displayed list is passed to the destination scene as a parameter.

4. Click Next.

   On the second page of the Drilldown wizard, specify the scene to jump to. Choose from the Location radio group to determine whether the drilldown action will jump to a scene in the currently selected dashboard, or in another visual project.

   • Select This dashboard to jump to a scene in the currently selected dashboard.

   • Select Another visual project to jump to a scene or a page in another visual project. The Visual project field becomes available. Use the ellipsis (...) to browse for the visual project you would like to jump to.

   • Click Retrieve Parameters to select from the available parameters of the specified visual project.
Note: To open the visual project in a new tab, select the **Open in new tab** check box.

5. Select the destination scene that you want the user to jump to from the **Scene** list box. If the scene does not exist, you will be asked to create one. Specify the viewpoint in the scene by selecting a viewpoint from the **Viewpoint** list box.

6. Click **Next**

   On the third page of the Drilldown wizard, specify the destination scene's parameter values. All the parameters that have been defined for the destination scene are listed in the Parameter field. These include parameters that apply only to the destination scene as well as the parameters for all the data columns that will be passed to the destination scene.

7. Click **Finish**.

   The click event navigation feature has been set for the layout object.

**Setting up information zooming:**

Information zooming is a navigation feature that presents a more detailed view of data as your user zooms in on a scene or a data point in a layout. Every layout object and scene has a default zoom level (100% zoom). Information zooming is only available in the Flash mode.

**About this task**

You can set up information zooming navigation for whole scenes and for individual layout objects. Information zooming on the layout object allows you to provide more query information on each data point as the user zooms in on a layout object. You use the same query results data for each zoom level. Information zooming on the scene level allows you to display an entirely different layout and query results data for each zoom level.

An example of using information zooming on a layout might be to design a carpool display with the locations of employees' homes, using a linear map layout object with three levels of detail. At the default zoom level (100%), the map displays markers to show the locations of each employee's home. At 200%, the map displays employee names. At 400%, the map displays employee names, phone numbers, and commute times. In this example, there are three levels of detail.

**Setting up information zooming for layout objects:**

When you set up information zooming for a layout you can provide more data from the same query as your user zooms in on a layout object.

**About this task**

All zoom levels in a layout object use the same query results data. For each layout object that you create, you can add multiple zoom levels. When you create a layout object a folder called **Level1** is created. This folder contains all the elements that will be displayed whenever the object is viewed at its initial, 100% or lower zoom level.

To incorporate information zooming, you add additional levels to the layout object. Each level that you add is numbered sequentially and added as a folder in the **Project Explorer** view. You add the objects that you want to display at a particular zoom level to the level folder. Each sequential level is displayed as the user changes to a higher zoom level.
To add a level of detail to a layout object:

**Procedure**

1. From the **Project Explorer** view, right-click the layout object's **DataTemplate** folder to which you want to add an additional level of detail. Select **Insert Level of Detail** from the pop-up menu.

2. An **Insert Level of Detail** message box is issued. This message asks how you want to initialize the new level. Select **Yes** to create a duplicate of the previous level. All the objects and their properties from the last level that appear in the folder are copied to the new level. Select **No** to create a blank level.

3. A folder for the new level is added to the layout object's folder in the **Project Explorer**. A new canvas labeled with the level name is presented in the editor. If you selected to model the new level on a previous level, all of the objects appear in the new level. If you selected to have a blank level, the canvas is empty.

4. After creating the new level, you can insert or delete objects, and edit the object properties. You can use any of the query results data that is available for the layout object.

5. When you create a new level, the zoom percentage that will trigger the level to be displayed is automatically assigned. The default value for each added level is twice the amount of the previous level. For example, the zoom level is 100% for the first level, 200% for the second level, and 400% for the third level. You can modify this value in one of two ways:
   
   - Edit the level's **MinimumZoom** property in the **Properties** view.
   - From the **Project Explorer** view, right-click the **Data Template** folder for the level whose zoom percentage you want to change. Select **Edit Level Transitions** from the pop-up menu. In the **Level of Detail Transition Editor** dialog box, specify zoom levels for each level of detail transition point.

**Results**

**Note:**

When you compose queries for layouts, include as many columns as you think might be valuable to users. You can display only the most important or highest-level column information in the default level of detail, and then add a level of detail to reveal more column information.

Design your data template and levels of detail at the same zoom level as they will appear to the user. When you insert text into a data template or a scene, the font size is set to scale to the zoom level. If you want the font size to remain constant regardless of the zoom level, change the **FixedSize** property for the **Font** object to **True** in the **Properties** view.

You can set up information zooming for a scene and for the layouts within the scene. In this case, if there are duplicate zoom levels, a scene level takes priority. For example, if your layout has the following four zoom levels; level 1 will display at the zoom percentage of 100%; level 2 will display at the zoom percentage of 125%; level 3 will display at the zoom percentage of 150%; level 4 will display at the zoom percentage of 200% and your scene has a level 2 that will display at 200%, then when your user switches to a 200% zoom level, the level 2 scene will be presented and not the level 4 object.
Setting up information zooming for scenes:

When you set up information zooming for a scene you can provide more data in different layouts and from multiple queries as your user zooms in on a scene.

About this task

For each scene that you create, you can add multiple zoom levels. When you create a scene a folder called Level1 is created. This folder contains all the elements that will be displayed whenever the scene is viewed at its initial, 100% or lower zoom level.

To incorporate information zooming, you add additional levels to the scene. Each level that you add is numbered sequentially and added as a folder in the Project Explorer view. You add the objects that you want to display at a particular zoom level to the level folder. Each sequential level is displayed as the user changes to a higher zoom level.

For example, you might create a second level of detail in the scene and include all the same objects in as the first level of detail, but with a different query used for the layout.

To add a level of detail to a scene:

Procedure

1. From the Project Explorer view, right-click the scene folder to which you want to add an additional level of detail. Select New > Level of Detail from the context menu.

2. An Insert Level of Detail message box is issued. This message asks how you want to initialize the new level. Select Yes to create a duplicate of the previous level. All the objects and their properties from the last level that appears in the folder are copied to the new level. Select No to create a blank level.

3. A folder for the new level is added to the scene's folder in the Project Explorer view. A new canvas labeled with the level name is presented in the editor. If you selected to model the new level on a previous level, all of the objects appear in the new level. If you selected to have a blank level, the canvas is empty.

4. After creating the new level, you can insert or delete objects, and edit the object properties.

5. When you create a new level, the zoom percentage that will trigger the level to be displayed is automatically assigned. The default value for each added level is twice the amount of the previous level. For example, the zoom level is 100% for the first level, 200% for the second level, and 400% for the third level. You can modify this value in one of two ways:
   • Edit the level’s MinimumZoom property in the Properties view.
   • From the Project Explorer view, right-click the Scene folder for the level whose zoom percentage you want to change. Select Edit Level Transitions from the pop-up menu. In the Level of Detail Transition Editor dialog box, specify zoom levels for each level of detail transition point.

Results

Note:
Design your scene and levels of detail at the same zoom level as they will appear to the user. When you insert text into a data template or a scene, the font size is set to scale to the zoom level. If you want the font size to remain constant regardless of the zoom level, change the FixedSize property for the Font object to True in the Properties view.

Create viewpoints to direct user attention to a particular level of detail. Using a jump to a viewpoint, you can allow users to go directly to the area in the scene and the level of the detail that contains information.

You can set up information zooming for a scene and for the layouts within the scene. In this case, if there are duplicate zoom levels, a scene level takes priority. For example, if your layout has the following four zoom levels; level 1 will display at the zoom percentage of 100%; level 2 will display at the zoom percentage of 125%; level 3 will display at the zoom percentage of 150%; level 4 will display at the zoom percentage of 200% and your scene has a level 2 that will display at 200%, then when your user switches to a 200% zoom level, the level 2 scene will be presented and not the level 4 object.

Inserting viewpoints:

Insert viewpoints in the navigation to isolate a particular area of a scene at a particular zoom level. Viewpoints define what part of a scene a viewer sees, and at what magnification.

About this task

You can use viewpoints as targets in jumps between scenes and associate them with a particular zoom level. For example, a scene might present a dashboard map indicating factory locations. You can insert and define viewpoints for each region of the dashboard, thereby allowing users to move quickly to the location and magnification for the region they are interested in.

In addition to providing a navigation tool, named viewpoints can be the target of an event action. For example, you can specify the display of a particular viewpoint if a user double-clicks an object.

To insert a viewpoint, perform the following steps:

Procedure

1. Open the New Viewpoint window in one of the following ways:
   • In the Canvas mode in the visual dashboard editor, select Insert > Insert Viewpoint to open the New Viewpoint window.
   • From the Project Explorer view, right-click on the Locals node for the scene to which you want to add a viewpoint and select Insert Viewpoint from the pop-up menu.
   • From the Project Explorer view, right-click on the scene for which you want to create the viewpoint and select New > Viewpoint

   The New Viewpoint window opens.

2. Type a name for the viewpoint in the Name field.
3. Type the location of the x-coordinate in the X location field.
   The X-coordinate specifies the distance from the center of the scene along the horizontal axis. The default value is 0.
4. Type the value of the y-coordinate in the Y location field.
The y-location is the number of inches from the center of the display.
The Y-coordinate specifies the distance from the center of the scene along the
vertical axis. The default value is 0.

Note: The default unit of measurement used for the X and Y coordinates
depend on the regional standards specified for the client machine. You can
change the unit of measurement by typing its abbreviation after the value. For
more information about units of measure, see the reference documentation in
the product help.

5. Specify the zoom level of the viewpoint in the **Zoom level** list.
   The default value is 100%.

6. Click **Finish**.
   The New Viewpoint window closes.

**Results**

To modify a viewpoint:

1. In the **Project Explorer** view, locate the viewpoint that you want to modify and
double-click it. The Edit Viewpoint Properties window opens.
2. Type changes in the **X location** and **Y location** fields.
3. Specify a new zoom level in the **Zoom level** list.
4. Click **OK**.

Note:

You can restore the scene to the default viewpoint by pressing the **Home** key.

The default viewpoint (0,0, 100% zoom) is a helpful orientation point when you are
developing a scene. In a large scene, when you want to return to the center, you
can press the **Home** key to take you back to the default viewpoint.

You can set a viewpoint for a scene from the **Default Viewpoint** folder in the
**Scenes** folder. Right-click **Default Viewpoint** and select **Properties** to change the
default viewpoint properties or select **Insert Viewpoint** to create a viewpoint.

**Modifying viewpoints:**

You can modify the viewpoints.

**About this task**

To modify a viewpoint:

**Procedure**

1. Open the **Edit Viewpoint Properties** window in one of the following ways:
   • In the **Project Explorer** view, locate the viewpoint that you want to modify
     and double-click it.
   • In the **Project Explorer** view, right-click the viewpoint that you want to
     modify and select **Edit Viewpoint**.
     The Edit Viewpoint Properties window opens.
2. Type changes in the **X location** and **Y location** text boxes.
3. Specify a new zoom level in the **Zoom level** list.
4. Click OK.
   The Edit Viewpoint Properties window closes.
   The default viewpoint (0,0, 100% zoom) is a helpful orientation point when you
   are developing a scene.
   To restore the scene to the default viewpoint:
   a. Right-click in the editor view and select Jump to... The Jump to window
      opens.
   b. Select the default and click OK.
   c. From the Project Explorer view, right-click the default viewpoint and select
      Jump to Viewpoint from the pop-up menu.
   d. Press the Home key.

Results
You have modified the viewpoint.

Using embedded scene objects:
An embedded scene object is a container through which you can view another
scene or a part of it and then navigate to that scene.

About this task
The scene where you place the embedded scene object is considered the source
scene. The destination scene is the embedded scene. When you insert an embedded
scene object it appears in the Scenes folder in the Project Explorer view.

An embedded scene differs from a jump or a hyperlink in the following ways:
• You see the embedded scene from the source scene.
• You can pass parameter information from the source scene to the embedded
  scene.

You can use parameters to pass information from one scene to another scene using
the embedded scene object. For example, if a source scene displays a scatter chart
of automobile sales, you can define each point on the chart as embedded scene
object that contains a destination scene displaying sales data for a particular
manufacturer. To create this example, you pass the manufacturer name as a
parameter to the embedded scene using the embedded scene object.

Results
Embedded scene objects have several properties that you might want to modify,
including:
• ZoomPercentage. Sets the zoom percentage of the embedded scene as viewed
  through the embedded scene object. You might want to reduce the zoom so that
  you see more of the embedded scene.
• SceneCenter. Sets the position of the embedded scene object over the embedded
  scene. By default, this property is set to the center of the embedded scene. You
  can change this value to display another area of the embedded scene. For
  example, you might have several charts on one scene, but want to display only
  one of them through the embedded scene object.
• BorderType. Sets the style of the border around the embedded scene object. You
  can choose a border style, or choose no border.
Inserting a simple embedded scene object:

A simple embedded scene connects two scenes.

Before you begin

For example, a source scene might display an overview of five mutual funds, with small charts for each fund. Each of the five charts can be an embedded scene that when selected will show a full-size display of a fund.

About this task

To insert a simple embedded scene object in your visual dashboard scene:

Procedure
1. From the Project Explorer, open the source scene in the Canvas mode.
2. From the Palette view, click the EmbeddedScene object.
3. Click in the editor window to insert the embedded scene object. The EmbeddedScene Wizard opens.
4. Specify whether you want to use a scene from the current Visual Project or from a previously saved one.
   If you want to open a scene from another Visual Project select the corresponding check box and select a project.
   If you want to use a scene from the current project make sure that the Another Visual Project check box is cleared.
5. Select the name of the scene that you will embed from the Embedded scene list box.
6. Select the Navigate to embedded scene when clicked with the mouse check box to automatically generate an event action for the embedded scene object. The generated event action specifies that a mouse click the embedded scene object will take a user to the embedded scene. The Navigate to embedded scene when clicked with the mouse check box should be cleared if you want to specify a different event action for the embedded scene object.
7. Specify the location and zoom level of the embedded scene in the X location, Y location, and Zoom fields.
8. Click Finish. The EmbeddedScene Wizard closes and the new embedded scene is displayed in the editor window.

Inserting a parameterized embedded scene object:

A parameterized embedded scene object connects two scenes and passes information from one scene to another.

About this task

For example, if a source scene displays a scatter chart of automobile sales, you can define each point on the chart as an embedded scene object that contains a destination scene displaying sales data for a particular manufacturer. You pass the manufacturer name as a parameter to the destination scene using the embedded scene object.

Note: You can use a scene from another Visual Project. If you want to open a scene from another Visual Project check the corresponding check box, click the
ellipsis (...) and browse for the visual project you want to use. When you close the Open wizard, path to the selected Visual Project is displayed in the Another visual project field.

To create a parameterized embedded scene object:

**Procedure**

1. Select the scene that you want to embed in the embedded scene object. Choose Insert > Scene to create a new scene or open an existing scene by double-clicking it in the Project Explorer view. If this is a new scene, or you have to edit the scene, add the text and graphical elements.

2. Specify that the embedded scene will receive parameters. Expand the folder for the scene that you want to embed. Expand the Locals folder for the scene. Right-click the Parameters folder and select Insert Parameter. The Insert Parameter window opens.

3. Enter the name of the parameter in the Name field and select a data type for the parameter from the Data type list box. You can enter a description for the parameter in the Description field. Select the Has default value if you would like to specify a default value for the parameter. A field where you will specify the default value opens. Click the ellipsis (...) for help in specifying the default value. The button that is provided depends on the data type that is selected. Click Finish. The Insert Parameter window closes.

4. Select the source scene where you will place the embedded scene object. Select Insert > Scene to create a new scene or open an existing scene by double-clicking it in the Project Explorer view. The source scene is displayed in the editor.

5. Add an embedded scene object to the source scene. From the Containers palette in the Palette view, double-click EmbeddedScene. The EmbeddedScene Wizard opens.

6. Select the name of the scene that you will embed from the Embedded scene list box.

7. Select the Navigate to embedded scene when clicked with the mouse check box to automatically generate an event action for the embedded scene object. The generated event action specifies that a mouse click the embedded scene object will take a user to the embedded scene. The Navigate to embedded scene when clicked with the mouse check box should be cleared if you want to specify a different event action for the embedded scene object.

8. Specify the location and zoom level of the embedded scene in the X location, Y location, and Zoom fields.

9. Click Finish. The EmbeddedScene Wizard closes and the new embedded scene is displayed in the editor window.

10. Select or create an object that can receive user input, for example a combo or list box.

11. Set the parameter value that will be passed to the embedded scene. In the Project Explorer view, expand one or more source scene folders to expose the embedded scene object. Expand the embedded scene object's folder. You will see the name of the embedded scene. Click the name of the embedded scene. The parameters that you have specified for the embedded scene are listed in the Properties view. Any default values are also listed. You can specify the values in the Properties view. Double-click the Value field to open the Expression Designer. The Expression Designer will help you specify the value for the parameter. If you used a list to accept user input then you must select a property value of the list to include in the parameter value.
Setting up event action navigation:

By assigning an event to a scene object or to any graphic object that can be inserted in a scene, you set up the object to be responsive to a user's actions as they navigate the dashboard.

Events are actions that are made by the user of the dashboard. Events include user actions such as a mouse click, mouse movements, or the use of keyboard keys. At the same time that you assign an event to an object, you also specify additional actions that will be triggered when the event occurs. Additional actions that you can define for an event include jumping to a new scene, setting a parameter value or issuing a message. As a user navigates the dashboard, when an event that you have assigned to an object occurs (such as a click on a button object), the additional actions that you have defined for the event are triggered.

The following tasks describe assigning events to objects and specifying the actions that will be triggered when the event occurs:

Assigning events and setting up event actions:

You use the Events view to assign an event to an object and set up the event actions.

About this task

The Events view displays the events tree with the events that can be assigned to the objects of a visual project and the list of event actions that are defined for the events.

You define event actions to objects to provide scene interactivity and navigation. As a user navigates your dashboard, mouse actions trigger predefined events, such as jumps between scenes, executing of SQL statements, or the launching of other applications.

To assign an event to an object and set up event actions:

Procedure

1. In most cases, the Events view opens with the Visual Designer perspective. If the Events view is not open, select Window > Show Views > Events to open the Events view.
2. Select an object in your visual project (either in the scene editor or in the Project Explorer view). The events that can be assigned to the selected object are displayed in the events tree in the upper pane of the Events view.
3. You can add an event to the selected object in one of the following ways:
   • In the events tree, double-click the event that you want to assign to the object. The [objectname] Behavior window opens.
   • In the events tree, select the event that you want to assign to the object and click the Add New Action button from the lower pane of the Events view. The Add New Action window opens. Then go to step 5 of the procedure.

Note: To quickly find the required event in the events tree, you can type the name of the event in the filter text field.
4. The event that will be assigned to the object is listed in the Event list box. You can select a different event from the drop-down list of available events. The event actions that were defined for the event are listed in the Actions list. To
add an action, click the Add icon. The Add New Action window opens. The actions that can be triggered by the event are displayed in the actions tree.

5. In the actions tree, select an action that you want to add to the event. You are prompted for additional information depending on the action that you choose.

**Note:** To display only event actions that are available for the selected event, ensure that the Hide disabled actions check box is selected.

To quickly find the required action in the actions tree, enter the action name in the filter text field.

6. The event action that you define is listed in the Action cell of the list box. For each action you can also specify condition and description.

The condition that you specify must be met before the action will occur. To specify a condition, click the Condition cell of the list box. Type an expression or click the ellipsis (...) to open the Expression Designer. You can use the Expression Designer to help you create the conditional expression.

Specify the description so that users can easily determine the purpose for which the action is applied.

7. You can specify that an event will trigger a series of multiple actions. Repeat the add process for each action that you want to add. Each action that you define is listed in the Actions list box. The actions that you define for an event are performed in the same order as they are listed in the Actions list box. You can use the Insert after selected item and Insert before selected item buttons to position new actions in the list. You can use the Move Action Up in List and Move Action Down in List icons in the toolbar to move the actions in the list to new positions. You can use the Copy Action and Paste Action icons in the toolbar to create duplicate actions.

8. You can delete any action that has been defined for the event. Select one or more actions from the Actions list box. Click the Remove Selected Actions from List icon. The action is deleted.

9. You can view or modify the details of an action. Select the action from the Actions list box. Click the Edit Selected Action icon. Depending on the action that you selected, a unique window opens that shows the details of the action. You can make changes to the details and save the changes.


**Results**

The event to which you have added one or more actions is shown in bold font in the events tree in the upper pane of the Events view. The list of actions added to the event is displayed in the lower pane of the Events view.

You can assign more than one event to an object. To assign another event to the object, select the event and repeat this process.

*Assigning jump events to objects in your visual dashboards:*

Use the Jump to new location event action to assign and define jump events to objects in your visual projects. At runtime, jump events take the user from an object in the dashboard or application to another scene or viewpoint of a scene or to a visual report.
Before you begin

You must create both the source and the destination dashboard scenes or viewpoints before creating the jump event between them. Additionally, if the jump event you are defining takes the user to a Visual Report, you must create the report before creating the jump event to that report.

About this task

Jumps can carry context information to the new location. For example, you can specify a scene parameter to be set at jump time and base the parameter on a calculated value or category, such as store location. Thus, a destination scene showing sales revenues can be based on store location and display different data depending on what context the user is coming from.

Note: You must create both the source and the destination project scenes or viewpoints before you can create a jump between them. You must also create a visual report before you can create a jump to the visual report.

To assign and define a jump event for an object in your visual project, perform the following steps:

Procedure

1. Select the object in the editor window and double-click the event that you want to trigger the jump from the Events view. The [objectname] Behavior window opens.
2. Click the Add New Action icon. The Add New Action window opens.
3. Select Jump to new location in the actions tree.

   Note: To quickly find the required event action, type the name of the action in the filter text field.
4. Click Next. The Jump to new location action parameters page opens.
5. Specify whether the destination of the jump will be within the current visual project or to another. Select This project to specify that the destination for the jump will be to a scene or viewpoint within the current project. Select Another visual project to specify that the destination for the jump will be another visual project.
6. If you selected This project, the Scene and Viewpoint fields become available. From the Scene list box, select the destination scene name. From the Viewpoint list box, select a viewpoint. If there are no viewpoints in the scene, the default is used.
7. If you selected Another visual project, the Visual project field becomes available. Specify the name of the destination visual project in the Visual project field. Click the ellipses button to search for and open a project from a repository or a file. The Open window opens. Navigate to and select the visual project that you want the event action to jump to and click Finish. The Open window closes.
8. Optional: If the visual project that you selected has parameters, click the Retrieve Parameters button.

   Note: The parameters that must be set when the event occurs are listed.
9. Specify a value for each of the parameters in the Value cell for each parameter.
10. From the **Open in** area select the required option:
   a. **Same tab** to open another visual project in the current tab.
   b. **New tab** to open another visual project in a new tab in DataQuant for Workstation or in a new browser tab in DataQuant for WebSphere.
   c. **New window** to open another visual project in a new tab in DataQuant for Workstation or in a new browser window in DataQuant for WebSphere.

11. Click **Finish**. The **Jump to new location** event action is added. The Add New Action window closes. Control returns to the [objectname] Behavior window.

12. From the [objectname] Behavior window you can add conditional expressions to the jump to a new location action. You can define multiple jump to new location actions each with a unique condition. Once a jump to a new location does occur, any subsequent actions defined for the object will not occur.

13. From the [objectname] Behavior window, click the **Apply** button to apply all actions that have been added for the event. Click **OK** to close the [objectname] Behavior window.

*Setting global and scene parameter values:*

Use the **Set values** action to set global and scene parameter values.

*About this task*

When an event occurs, you can set global and scene parameters to new values. The new values can be constants, functional expressions of data values, or other parameters (global, scene, or event parameters). You must create the parameters that will be used.

To set parameter values when an event occurs:

*Procedure*

1. Select the object in the editor window and double-click the event that you want to trigger the jump from the **Events** view. The [objectname] Behavior window opens.
2. Click the **Add New Action** icon. The Add New Action window opens.
3. Select **Set values** in the actions tree.

   **Note:** To quickly find the required event action, type the name of the action in the filter text field.

4. Click **Next**. The Set values action parameters page opens.
5. Click in the **Parameter** field and select a parameter name from the list of available parameters. All the parameters that you have created and are available to the scene are listed.
6. Click in the corresponding **Value** field and select a value from the list box. The parameter value must match the parameter type.
7. To add a parameter, click the **Add New Entry** button. Type the name of the new parameter in the **Parameter** field. Specify the value for the parameter in the **Value** field.
8. To delete a parameter, select the parameter and click the **Delete** icon.
9. Click **Finish**. The **Set values** action is added. The Add New Action window closes.
10. From the [objectname] Behavior window, click the **Apply** button to apply all actions that have been added for the event.
11. Click OK to close the [objectname] Behavior window.

ExternalEvent events:

ExternalEvent events occur when a user sends some event to an embedded scene object by using the Send to Embedded Scene event action.

You can assign ExternalEvent events to the following objects:

- scene objects

You can incorporate ExternalEvent events in scene objects to trigger different event actions. For more information about event actions, see “Assigning events and setting up event actions” on page 269 topic.

Adding a Navigate to embedded scene event action:

Use the Navigate to embedded scene event action to move the user to a scene or viewpoint of a scene that has been embedded in an EmbeddedScene object.

About this task

Embedded scenes are added and set up automatically when a user clicks directly the embedded scene object. This is the default behavior.

You might want to add a Navigate to embedded scene event action manually for the following scenarios:

- To specify an event (such as a click) on a different object (such as a button) to trigger entering the embedded scene of an EmbeddedScene object.
  For example, you can insert a Button object in a dashboard scene and specify that a specific embedded scene (that has been inserted in the same dashboard scene) will be entered when a user clicks the button.
- To add an event (in addition to the Click event) to trigger entering the embedded scene of an EmbeddedScene object.
- To specify an event (other than the Click event) to trigger entering the embedded scene of an EmbeddedScene object.
  For example, you might want a DblClick event to trigger entering the embedded scene of an EmbeddedScene object.

To add and define a Navigate to embedded scene event action to an object:

Procedure
1. Select the object in the editor window. Double-click the event that you want to trigger the action from the Events view. The [objectname] Behavior window opens.
2. Click the Add icon. The Add New Action window opens.
3. Select Navigate to embedded scene in the actions tree.

Note: To quickly find the required event action, type the name of the action in the filter text field.
4. Click Next. A window requesting the embedded scene action parameters opens.
5. Select an embedded scene object from the Embedded scene list of available embedded scene objects. Each embedded scene object that you have inserted in the current source scene is listed.
6. Select a viewpoint from the Viewpoint on [scenename] list of available viewpoints for the embedded scene. Each viewpoint object that you have created for the embedded scene is listed. If there are no viewpoints in the scene, the default viewpoint is listed.

7. Click Finish. The Navigate to embedded scene event action is added. The Add New Action window closes.

8. From the [objectname] Behavior window, click Apply to apply all actions that have been added for the event. Click OK to close the [objectname] Behavior window.

Adding a Navigate back event action:

Use the Navigate back event action to return a user from the current object to the previous scene or to the previous viewpoint of a scene. For example, you can specify that when a user performs an event (such as a mouse click) on an object (such as a button), the result is that they are sent back to the previous scene.

About this task

To add a Navigate back event action to an object:

Procedure
1. Select the object in the editor window and double-click the event that you want to trigger the jump from the Events view. The [objectname] Behavior window opens.
2. Click the Add New Action icon. The Add New Action window opens.
3. Select Navigate back in the actions tree.
   
   **Note:** To quickly find the required event action, type the name of the action in the filter text field.
4. Click Finish. There are no additional parameters required for the Navigate back action. The Navigate back action is added. The Add New Action window closes.
5. From the [objectname] Behavior window, click Apply to apply all actions that have been added for the event. Click OK to close the [objectname] Behavior window.

Adding an Execute SQL statement event action:

Use the Execute SQL statement action to run an SQL statement after a specified event occurs.

About this task

To add an Execute SQL statement event action to an object:

Procedure
1. Select the object in the editor window. The [objectname] Behavior window opens.
   
   From the Events view, double-click the event that you want to trigger the Execute SQL statement action.
2. Click the Add icon. The Add New Action window opens.
3. Select Execute SQL statement in the actions tree and click Next.
Note: To quickly find the required event action, type the name of the action in the filter text field.

4. Enter the SQL statement in the SQL statement field.

The SQL statement can contain parameters. If you want to include global or scene parameters in your SQL statement, you must enter the SQL statement as an expression, much as you would enter an expression into a property value.

For example, to run the following statement:

```
Update MyTable Set Shipped = 1
Where CustomerID = [value of g_nCustID global variable]
```

You would enter the following text in the SQL Statement field:

```
=Update MyTable Set Shipped = 1 Where CustomerID = " + g_nCustID
```

5. Optional: If the SQL statement affects the content of a scene viewed by your user, be sure to select the Refresh the display on completion check box.

6. Optional: Select Display a message box indicating the result of the operation if you want to display a message about the success or failure of the execute SQL statement.

If you decide to display a message box indicating the result of the operation, enter the text of the message that you want to display in both the Success field and the Failure field.

Note: Use $ (1) to include the system output message as part of the success or failure message text.

7. Select Save the success or failure (and any return value) in a specified result parameter to save the result of the executed SQL statement in either a local or global parameter.

When you select this check box the Result parameter lists become available.

The local and global parameters that have been created for your dashboard are listed in the Result parameter drop-down lists.

Select the local or global parameter that you want to use to store the result information. The results will be stored as follows:

- If the SQL statement returns one or more values, the first column in the first row is placed in your selected global or scene parameter. For example, the statement `select count(*) from MyTable` will place the count in your selected parameter. The statement `select A, B, C from MyTable` will place the value of A from the first row in your selected parameter.

- If the statement does not produce results, the success or failure of the statement execution is placed in your selected parameter. For example, the statement `update MyTable Set A = 3 where CustomerID = 1` will place true or 1 in your selected parameter if the statement executed successfully, or false or 0 if the statement failed to execute. When this parameter is initially created, it has a data type of Boolean. You can modify the name and data type of the global parameter.


9. From the [objectname] Behavior window, click Apply to apply all actions that have been added for the event. Click OK to close the [objectname] Behavior window.

Results

You have added an Execute SQL statement event action to an object.
Adding an Execute procedure command event action:

Use the **Execute procedure** action to run a procedure command after the specified event occurs

About this task

To add an **Execute procedure** event action to an object:

Procedure

1. Select the object in the editor window. Double-click the event that you want to trigger the action from the **Events** view. The [objectname] Behavior window opens.
2. Click the **Add** icon. The Add New Action window opens.
3. Select **Execute procedure** in the actions tree and click **Next** to open the Execute procedure action parameters page.

   **Note:** To quickly find the required event action, type the name of the action in the filter text field.
4. Use the **From Repository** field to enter the procedure statement name and location.
5. Select the **Hide procedure result windows** check box to hide the results window after running the procedure statement.
6. Select the **Refresh display on completion** check box to refresh the dashboard display after running the procedure statement.
7. Select the **Display a message box indicating the result of the operation** check box to display the success or failure of the run procedure statement. If you check this check box:
   - Type the text that you want displayed upon a successful run in the **Success** field. You can use $(1) in your message to include the system output message as part of the success message text.
   - Type the text that you want displayed upon a failed run in the **Failure** field. You can use $(1) in your message text to include the system output message as part of the failure message.
8. Check the **Save the success or failure (and any return value) in a specified result parameter** check box to save the result of the executed procedure statement in either a local or global parameter. When this check box is selected the **Result parameter** lists become available. The local and global parameters that have been created for your dashboard are listed in the **Result parameter** drop-down lists. Select the local or global parameter that you want to use to store the result information. The results will be stored as follows:
   - If the procedure statement returns one or more values, the first column in the first row is placed in your selected global or scene parameter. For example, the statement `select count(*) from MyTable` will place the count in your selected parameter. The statement `select A, B, C from MyTable` will place the value of A from the first row in your selected parameter.
   - If the procedure statement does not produce results, the success or failure of the statement execution is placed in your selected parameter. For example, the statement `update MyTable Set A = 3 where CustomerID = 1` will place true or 1 in your selected parameter if the statement executed successfully, or false or 0 if the statement failed to execute. When this parameter is initially created, it has a data type of Boolean. You can modify the name and data type of the global parameter.
9. Click **Finish**. The **Execute procedure** action is added. The Add New Action window closes.

10. From the [objectname] Behavior window, click **Apply** to apply all actions that have been added for the event. Click **OK** to close the [objectname] Behavior window.

*Adding an Execute JavaScript event action:*

Use the **Execute JavaScript** action to run JavaScript after the specified event occurs. For example, you can run JavaScript to open a new window or start the system print dialog.

*About this task*

To add an **Execute JavaScript** event action to an object:

**Procedure**

1. From the **Events view**, select the object in the editor window and double-click the event that you want to trigger the **Execute JavaScript** action. The [objectname] Behavior window opens.
2. Click the **Add New Action** icon. The Add New Action dialog opens.
3. Select **Execute JavaScript** in the actions tree and click **Next** to open the Execute JavaScript action parameters page.
   
   **Note:** To quickly find the required event action, type the name of the action in the filter text field.
4. Type JavaScript commands in the **JavaScript** field and click **Finish**. The JavaScript command action is added. The Add New Action window closes.
5. From the [objectname] Behavior window, click **Apply** to apply all actions that have been added for the event.
6. Click **OK** to close the [objectname] Behavior window.

*Results*

You have added an Execute JavaScript event action to an object.

*Adding a Print scene event action:*

Use the **Print scene** to add a print event action. For example, you can dictate that when the user clicks a **Button** object on a particular scene, the result will be to print the values that are displayed.

*About this task*

To incorporate a **Print scene** event action in an object:

**Procedure**

1. From the **Events view**, select the object in the editor window and double-click the event that you want to trigger the **Print scene** action. The [objectname] Behavior window opens.
2. Click the **Add New Action** icon. The Add New Action dialog opens.
3. Select **Print scene** in the actions tree.
Note: To quickly find the required event action, type the name of the action in the filter text field.

4. Click Next.
   The Print scene action parameters page opens.
   If, when printed, the scene does not fit within the print area, select the Shrink to page check box.

5. Click Finish.
   The Print scene action is added and the Add New Action window closes.

6. From the [objectname] Behavior window, click Apply to apply all actions that have been added for the event.

7. Click OK to close the [objectname] Behavior window.

Adding a Send Email event action:

Use the Send Email event action to send a scene to an email recipient after the assigned event occurs. For example, you might incorporate this action when the user clicks Preview on a particular scene, so as a result the scene is sent to an email recipient.

About this task

To add a Send email event action to an object:

Procedure

1. From the Events view, select the object in the editor window. Double-click the event that you want to trigger.

2. In the [objectname] Behavior window, click the Add icon. The Add New Action window opens.

3. Select Send Email in the actions tree.

   Note: To quickly find the required event action, type the name of the action in the filter text field.

4. Click Next. A dialog requesting the Send Email parameters opens.

5. Specify the address that the email will be sent to in the To field.

6. Specify the address that a carbon copy of the email will be sent to in the Cc field.

7. Specify the subject of the email in the Subject field.

8. Enter the body text of the email in the Message field.

9. Specify the address that will send the email in the From field.

10. To specify SMTP server parameters, click SMTP settings. In the SMTP Settings window, specify whether to use a server predefined by administrator or a custom one.

   a. If you decide to use a predefined SMTP server, select the required server from the list of predefined servers.

   b. If you decide to use a custom server, specify server parameters including its address and port number. If the SMTP server requires authentication, select Use SMTP authentication, and then specify user name and password. If it is necessary to specify additional access preferences, click Advanced to open the Advanced SMTP Settings window where you can add required keywords supported by javamail. Click OK.

Click OK to close the SMTP Settings window.
11. To include an attachment with the email, create a local or global parameter that specifies the file path and file name of the item that you want to send.
12. Select whether the file path is taken from a global or local parameter and what parameter is used in the Take file path from parameter drop-down lists.

Note: Valid parameter types are FilePath, Literal, and Text.
13. Click Finish. The Send Email action is added. The Add New Action window closes.
14. From the [objectname] Behavior window, click the Apply button to apply all actions that have been added for the event.
15. Click OK to close the [objectname] Behavior window.

Adding a Send event to embedded scene action:

Use the Send Event to Embedded Scene action to initiate a specific event in an embedded scene

About this task

The Send Event to Embedded Scene action generates an event with a specified ID and sends it to the specified embedded scene.

To add a Send event to embedded scene event action to an object:

Procedure
1. Select the object in the editor window. Double-click the event that you want to trigger the action from the Events view. The [objectname] Behavior window opens. Click the Add icon. The Add New Action window opens.
2. Select Send Event to Embedded Scene in the actions tree.

Note: To quickly find the required event action, type the name of the action in the filter text field.
3. Click Next. A dialog requesting the Send Event to Embedded Scene action parameters opens.
4. Specify at least one embedded scene in the Target scenes field.
5. Optional: In the Result options section, specify the global parameter to be used as an external event identifier in the target scene.
   You can add a new parameter by clicking the Add New Parameter.

Note: The ExternalEventID global parameter is used by default.
6. Select the parameter that you want to use from the Result parameter drop down lists.
8. From the [objectname] Behavior window, click the Apply button to apply all actions that have been added for the event.
9. Click OK to close the [objectname] Behavior window.
10. From the Project Explorer view, select a scene connected with the target embedded scene.
11. From the Events view select the ExternalEvent item and double-click it. In the [objectname] Behavior window, set up the event actions.

Consider the example:
You have the scenes: **Scene1** and **Scene2** and the buttons: **Button1** and **Button2** on **Scene1**.

1. Add the following actions to the buttons: **Button1** action is **ExportToPDF** and **Button2** action is **SendEmail**.
2. Set the values of **Event ID**, type any string there, for example, **Button1 Event ID** is **Btn1ExportToPDF** and **Button2 Event ID** is **Btn2SendEmail**.
3. Add different **ExternalEvent** actions to the **Scene2** and set the following conditions of the **ExternalEventID** parameters:

<table>
<thead>
<tr>
<th>Action</th>
<th>Condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>ExportToPDF</td>
<td>=ExternalEventID = 'Btn1ExportToPDF'</td>
</tr>
<tr>
<td>SendEmail</td>
<td>=ExternalEventID = 'Btn2SendEmail'</td>
</tr>
</tbody>
</table>

These actions will be applied to the content of the **Scene2**.

Click **OK** to close the [objectname] Behavior window.

Adding an Execute shell command event action:

Use the **Execute shell command** action to run a shell command after a specified event occurs. The **Execute shell command** event action allows you to run a shell command like that performed using the operating system Run command from the Start menu.

About this task

For example, you can run a shell command to open Notepad or an email program.

To add an **Execute shell command** event action to an object:

Procedure

1. Select the object in the editor window and double-click the event that you want to trigger the jump from the **Events** view. The [objectname] Behavior window opens.
2. Click the **Add New Action** icon. The Add New Action window opens.
3. Select **Execute shell command** in the actions tree.

   **Note:** To quickly find the required event action, type the name of the action in the filter text field.

4. Click **Next**. The Execute shell command action parameters page opens.
5. Type the command in the **Command** box or click **Browse** to browse and select a file that contains the commands.

   **Note:** You can use expressions to specify commands. For example, type 
   `"%SystemRoot%\notepad.exe " + Var1` to open a file, the path to which is stored in the Var1 parameter, in Notepad.
6. Click **Finish**. The **Execute shell command** action is added. The Add New Action window closes.
7. From the [objectname] Behavior window, click the **Apply** button to apply all actions that have been added for the event.
8. Click **OK** to close the [objectname] Behavior window.
Adding a Show message event action:

Use the Show message event action to display a message, save the user response, and use it in other actions.

About this task

To add a Show message event action to an object:

Procedure
1. Select the object in the editor window and double-click the event that you want to trigger the jump from the Events view. The [objectname] Behavior dialog opens.
2. Click the Add New Action icon. The Add New Action dialog opens.
3. Select Show message in the actions tree and click Next.

   Note: To quickly find the required event action, type the name of the action in the filter text field.
   The Show message action parameters page opens.
4. Type the text of the message that you want to issue in The text of message field.
5. Specify where the message is issued by using the Display type radio buttons. Select Message box to display the message in a standard message box. Select Output view to display the message in the Output view of DataQuant for Workstation.
6. Select the severity level of the message from the Logging level list.
7. If you want to specify extra message box parameters, click Next, otherwise go to the step 10.
8. In the Message Box area, specify the following parameters:
   a. In the Title field, enter the title that you want to use for the message box. Double-click the field to open the Expression Designer.
   b. From the Buttons list, select the button combinations that you want to add to the message box. The indexes of the selected buttons can be saved to parameters.

      Tip: The indexes of the buttons begin with 1.

      Tip: If a user closes the message box, the index of the last button is received.
   c. From the Icon list, select the icon that you want to be added to the message box.
9. If you want to save the button index of the button that is clicked by a user, perform the following steps:
   a. To save the index to the existing parameter, in the Result Options area, select the Save result to parameter check box
   b. From the Result parameter list, select the parameter that you want to use.
   c. To add a parameter, click Add New Parameter button.
10. Click Finish. The Show message action is added. The Add New Action dialog closes.
11. From the [objectname] Behavior dialog, click Apply to apply all actions that was added for the event.
12. Click **OK** to close the [objectname] Behavior dialog.

**Example**

If you create a message with the Yes/No/Cancel combination of buttons, the following indexes can be received:

*Table 27. Buttons and their indexes.*

<table>
<thead>
<tr>
<th>Clicked Button</th>
<th>Received Index</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yes</td>
<td>1</td>
</tr>
<tr>
<td>No</td>
<td>2</td>
</tr>
<tr>
<td>Cancel</td>
<td>3</td>
</tr>
<tr>
<td>A user closes the message box</td>
<td>3</td>
</tr>
</tbody>
</table>

**Adding an Export to Excel event action:**

Use the **Export to Excel** action to export a specified event to Excel.

**Before you begin**

See system requirements in the Installing and Managing guide for information about the version of Excel required to support the **Export to Excel** feature.

**About this task**

To add an **Export to Excel** event action to an object:

**Procedure**

1. From the Events view, select the object in the editor window and double-click the event that you want to trigger. The [objectname] Behavior window opens.
2. Click the **Add New Action** icon. The Add New Action window opens.
3. Select **Export to Excel** in the actions tree and click **Next** to open the Export to Excel action parameters page.

   **Note:** To quickly find the required event action, type the name of the action in the filter text field.

4. Specify the source component name in the **Data Template** field and click the ellipsis (...).
   
   The Select data template window opens.
   
   Select the data source template you want to use and click **OK** to close the window.
   
   The system fills in the highest data level and populates the properties and columns entries in the **Export data** text block.

5. Use the **Move Up** and **Move Down** icons in the toolbar to move the entries in the list to new positions.
6. You can delete any entry that has been defined for the export columns. Select an entry from the **Export columns** list box. Click the **Delete Entry** icon. The entry is deleted.
7. In the **Export type** radio group, select one of three radio buttons:
   - **Open.** It opens **Save As** dialog. Enter the name of the file in the **File name** field to save it.
• **Save to a temp file.** It saves the generated file in a temporary directory. The path to this file is recorded in a global or local parameter. You specify the parameter in the bottom of the window.
  – Select either Global parameters or Local parameters from the Result parameter drop-down list.
  – Click Add New Parameter. The Insert Parameter window opens.
  – Enter a name for the parameter and its description in the Name and Description fields.
  – Select a parameter type. The valid types are FilePath, Literal, and Text.
  – Select the Has default value check box and specify the full file path in the text field; for example, C:\DOCUME~1\<user>\LOCALS~1\Temp\<software_product_name>\reporter-<user>\pdf.

• **Save to a file.** It automatically saves the generated file in a specified root directory when the event occurs.

8. If you select Save to a file check box there are several ways of building an export path, depending on the directory which can be specified in the Root output directory field on the Server-Side File System page of the Preferences window:
  • When the output root directory is specified, the export directory of the file will be combined of the root output directory and the path typed in the Path field.
  • In DataQuant for Workstation when the output root directory is not specified, the export directory of the file will be a combination of the application working directory and the path typed in the Path field.
  • In DataQuant for WebSphere when the output root directory is not specified, the export directory of the file will be a combination of the temporal directory and the file name, the typed path is ignored.

9. From the Format type list, select the Excel file format in which you want to save the exported data. The available options are:
  • XLS Microsoft Excel 97-2003 (*.xls)
  • XLSX Microsoft Excel 2007 (*.xlsx)

10. To filter the available dashboard components, specify a string in the type filter text field. Only the components that have names that contain the specified string will be displayed in the Export data tree. If you selected Save to a temp file or Save to a file, the Result options section is active.

11. If you did not previously specify the directory parameter, select either Global parameters or Local parameters from the Result parameter drop-down list and click the Add New Parameter button. The Insert Parameter window opens.

12. Specify a necessary parameter and click Finish. The Insert Parameter window closes and control returns to the Add New Action window.

13. Click Finish. The Export to Excel action is added. The [objectname] Behavior window is updated.


Related tasks:

Setting the directory location preference for objects generated by Export actions
You can set preferences in DataQuant so that objects generated from various DataQuant actions are exported to a specific location on either the local file system or the server file system.
Adding a Set focus event action:

Use the Set focus event action to set the focus on any given control name parameter. For example, you might incorporate the Set focus event action so that when a user makes a selection, the result is to take the control name as the parameter on which the focus needs to be set.

About this task

To add a Set focus event action to an object:

Procedure

1. From the Events view, select the object in the editor window and double-click the event that you want to trigger. The [objectname] Behavior window opens.
2. Click the Add New Action icon. The Add New Action window opens.
3. Select Set focus in the actions tree.

   Note: To quickly find the required event action, type the name of the action in the filter text field.
4. Click Next. The Set focus action parameters window opens.
5. Specify the control name of the component in one of the following ways:
   • Type the name in the Component field;
   • Click Set from current level to choose the component.
     The Set component dialog opens. Select the component from the current level and click OK to close the Set component dialog.
6. Click Finish. The Set focus action is added. The Add New Action window closes.
7. From the [objectname] Behavior window, click the Apply button to apply all actions that have been added for the event.
8. Click OK to close the [objectname] Behavior window.

Results

You have added a Set focus event action to an object.

Adding a Reload scene event action:

Use the Reload scene event action to reload the scene after selecting a translation table language. For example, the user can select a Translation table language on a particular scene and have the selected language reflected when the user navigates to another scene.

About this task

To add a Reload scene event action to an object:

Procedure

1. Select the object in the editor window and double-click the event that you want to trigger the jump from the Events view. The [objectname] Behavior window opens.
2. Click the Add New Action icon. The Add New Action window opens.
3. Select Reload scene in the actions tree.
**Note:** To quickly find the required event action, type the name of the action in the filter text field.

4. Click **Finish**. The **Reload scene** action is added. The Add New Action window closes.

5. From the [objectname] Behavior window, click the **Apply** button to apply all actions that have been added for the event.

6. Click **OK** to close the [objectname] Behavior window.

**Adding an End session event action:**

Use the **End Session** event action to end the current session and close the application.

**About this task**

By adding a **End Session** event action to a visual dashboard, you can close the application from the project without having access to the application’s user interface. This is useful when you are accessing a visual dashboard from an embedded window.

To add an **End session** event action to an object:

**Procedure**

1. Select the object in the editor window and double-click the event that you want to trigger the jump from the **Events** view. The [objectname] Behavior window opens.

2. Click the **Add New Action** icon. The Add New Action window opens.

3. Select **End Session** in the actions tree.

   **Note:** To quickly find the required event action, type the name of the action in the filter text field.

4. Click **Finish**. The **End Session** action is added. The Add New Action window closes.

5. From the [objectname] Behavior window, click the **Apply** button to apply all actions that have been added for the event.

6. Click **OK** to close the [objectname] Behavior window.

**Adding an Export to PDF event action:**

Use the **Export to PDF** event action to export specific components of a visual dashboard to a PDF file.

**About this task**

The **Export to PDF** event action allows you to select specific components of a visual dashboard that you want to export to a PDF file, while excluding those that are unnecessary or extraneous. This is useful for situations where a visual dashboard has a recurring graphical object, like a header, that would take up unneeded space in a printed format.

**Restriction:** In the HTML5 mode, the content of Google map and map objects cannot be exported in this action.

To add an **Export to PDF** event action to an object:
Procedure

1. Select the object in the editor window and double-click the event that you want to trigger the jump from the Events view. The [objectname] Behavior window opens.

2. Click the Add New Action icon. The Add New Action window opens.

3. Select Export to PDF in the actions tree.

   Note: To quickly find the required event action, type the name of the action in the filter text field.

4. Click Next. An Export to PDF specific window opens.

5. In the Export type radio group, select one of three radio buttons:
   - Open. It opens Save As dialog. Enter the name of the file in the File name field to save it.
   - Save to a temp file. It saves the generated file in a temporary directory. The path to this file is recorded in a global or local parameter. You specify the parameter in the bottom of the window.
     - Select either Global parameters or Local parameters from the Result parameter drop-down list.
     - Click Add New Parameter. The Insert Parameter window opens.
     - Enter a name for the parameter and its description in the Name and Description fields.
     - Select a parameter type. The valid types are FilePath, Literal, and Text.
     - Select the Has default value check box and specify the full file path in the text field; for example, C:\DOCUME~1\<user>\LOCALS~1\Temp\<software_product_name>\reporter-<user>\pdf\..
   - Save to a file. It automatically saves the generated file in a specified root directory when the event occurs.

6. If you select Save to a file check box there are several ways of building an export path, depending on the directory which can be specified in the Root output directory field on the Server-Side File System page of the Preferences window:
   - When the output root directory is specified, the export directory of the file will be combined of the root output directory and the path typed in the Path field.
   - In DataQuant for Workstation when the output root directory is not specified, the export directory of the file will be a combination of the application working directory and the path typed in the Path field.
   - In DataQuant for WebSphere when the output root directory is not specified, the export directory of the file will be a combination of the temporal directory and the filename, the typed path is ignored.

7. To filter the available dashboard components, specify a string in the type filter text field. Only the components that have names that contain the specified string will be displayed in the Export data tree. If you selected Save to a temp file or Save to a file, the Result options section is active.

8. If you did not previously specify the directory parameter, select either Global parameters or Local parameters from the Result parameter drop-down list and click the Add New Parameter button. The Insert Parameter window opens.

9. Specify a necessary parameter and click Finish. The Insert Parameter window closes and control returns to the Add New Action window.
10. Select the parameter that you want to use from the Result parameter drop-down lists and click OK. The Add New Action dialog closes.

11. From the [objectname] Behavior window, click Apply to apply all actions that have been added for the event.

12. Click OK to close the [objectname] Behavior window.

Related tasks:

- Setting the directory location preference for objects generated by Export actions
  You can set preferences in DataQuant so that objects generated from various DataQuant actions are exported to a specific location on either the local file system or the server file system.

Adding a Create File event action:

Use the Create File to create a specific content while running a visual project and then use this content for different purposes, for example, sent it to mail.

About this task

To add an Create File event action to an object:

Procedure

1. Select the object in the editor window and double-click the event that you want to trigger the jump from the Events view. The [objectname] Behavior window opens.

2. Click the Add New Action icon. The Add New Action window opens.

3. Select Create File in the actions tree and click Next to open the Create File specific window.

   Note: To quickly find the required event action, type the name of the action in the filter text field.

4. In the Output type radio group, select one of three radio buttons:
   - Open. It opens Save As dialog. Enter the name of the file in the File name field to save it.
   - Save to a temp file. It saves the generated file in a temporary directory. The path to this file is recorded in a global or local parameter. You specify the parameter in the bottom of the window.
     - Select either Global parameters or Local parameters from the Result parameter drop-down list.
     - Click Add New Parameter. The Insert Parameter window opens.
     - Enter a name for the parameter and its description in the Name and Description fields.
     - Select a parameter type. The valid types are FilePath, Literal, and Text.
   - Save to a file. It automatically saves the generated file in a specified root directory when the event occurs.

5. If you select Save to a file check box you can build an export path in different ways, depending on the directory which can be specified in the Root output directory field on the Server-Side File System page of the Preferences window:
   - When the output root directory is specified, the export directory of the file will be combined of the root output directory and the path typed in the Name field.
• In DataQuant for Workstation when the output root directory is not specified, the export directory of the file will be a combination of the application working directory and the path typed in the Name field.

• In DataQuant for WebSphere when the output root directory is not specified, the export directory of the file will be a combination of the temporal directory and the file name, the typed path is ignored.

6. Specify a file name for the file you want to create in the Name field. You can also specify the file extension in this field. If you do not specify it, the file type is selected from the list of available ones:

   • application/octet-stream - *.bin
   • image/bmp - *.bmp
   • image/png - *.png
   • text/calendar - *.ics
   • text/plain - *.txt

Note: Specify a file extension if you want to create a file different from the listed ones.

7. In the File Content area, enter the content of the file you want to create in the Content field. For example, you want to add the text from TextBox1, enter the value: = TextBox1.Text.

8. Select the type from the Content type list or enter the any necessary value in the list. The available types for the content selection are:

   • application/octet-stream specifies a binary file that is associated with an application that opens this file.
   • image/bmp creates an image in *.bmp format.
   • image/png creates an image in *.png format.
   • text/calendar creates a document in iCalendar data format for representing and exchanging, calendaring and scheduling calendar information.
   • text/plain creates an ordinary sequential file readable as textual material without much processing.

9. If you did not previously specify the directory parameter, select either Global parameters or Local parameters from the Result parameter drop-down list and click the Add New Parameter button. The Insert Parameter window opens.

10. Specify a necessary parameter and click Finish. The Insert Parameter window closes and control returns to the Add New Action window.

11. Select the parameter that you want to use from the Result parameter drop-down lists and click OK. The Add New Action dialog closes.

12. From the [objectname] Behavior window, click Apply to apply all actions that have been added for the event.

13. Click OK to close the [objectname] Behavior window.

Adding a Refresh component event action:

Use the Refresh Component event action to refresh a component in a visual dashboard by invalidating a query cache. This event action reruns the query and the results are displayed in the refreshed component.
About this task

The Refresh Component event action is appropriate when querying related components only.

To add a Refresh component event action to an object:

Procedure
1. Select the object in the editor window and double-click the event that will trigger the jump from the Events view. The [objectname] Behavior window opens.
2. Click the Add New Action icon. The Add New Action window opens.
3. Select Refresh Component in the actions tree.

   Note: To quickly find the required event action, type the name of the action in the filter text field.
4. Click Next.

   A window requesting the parameters for the Refresh Component event action opens.
5. Specify one or more target components to refresh and click Finish.

   The Add New Action window closes.
6. Click OK to close the [objectname] Behavior window.

Results

You have added a Refresh Component event action.

The target component will be refreshed when the event action that you have specified occurs.

Adding an Invalidate query cache event action:

Use the Invalidate Query Cache event action to invalidate any query cache, if the caching of result set data is enabled.

About this task

Using this event action is appropriate in a scenario where you have changed the target query table, thereby invalidating any previous results, and these invalid results are still displaying in the related components. By implementing the Invalidate Query Cache event action, you redraw results in all the related components. This query is rerun and the valid results are displayed in the component.

To add an Invalidate query cache event action to an object:

Procedure
1. Select the object in the editor window and double-click the event that will trigger the jump from the Events view. The [objectname] Behavior window opens.
2. Click the Add New Action icon. The Add New Action dialog opens.
3. Select Invalidate Query Cache in the actions tree.
Note: To quickly find the required event action, type the name of the action in the filter text field.

4. Click Next.

A window requesting the parameters for the Invalidate Query Cache event action opens.

5. Specify one or more target components for which the query cache will be invalidated and click Finish.

The Add New Action window closes.

6. Click OK to close the [objectname] Behavior window.

Results

You have added an Invalidate Query Cache event action.

When the defined event action occurs, query cache in the target component is invalidated and the valid results are displayed in the component the next time it is refreshed.

Adding a Launch LOB event action:

Use the Launch LOB event action to open large object (LOB) data from an object of a visual dashboard.

About this task

LOB data is binary data of any type (for example, PDF files and the content of image files such as .gif or .jpeg files) that can be stored in a database table.

When run, the Launch LOB event action reads LOB values stored in a database table and performs an action associated with the value. The developer must specify the LOB data type in order for the application to launch the appropriate program and display the data.

Note: The Launch LOB event action can be applied to only those objects that are contained in the data template.

To add a Launch LOB event action, perform the following steps:

Procedure

1. Select the object in the editor window and double-click the event that will trigger the jump from the Events view. The [objectname] Behavior window opens.

2. Click the Add New Action icon. The Add New Action window opens.

3. Select Launch LOB in the actions tree.

    Note: To quickly find the required event action, type the name of the action in the filter text field.

4. Click Next. A window requesting the parameters for the Launch LOB event action opens.

    a. Select Data from the drop-down list or type the necessary expression.

        Note: The result of the expression must be the name of the column containing LOB data.
Select Data from the drop-down list or type the necessary expression. The result of the expression must be the name of the column containing LOB data.

b. Specify the data type by typing the necessary expression.

5. Click Finish.
   The Add New Action window closes.

6. From the [objectname] Behavior window, click Apply to apply all actions that you have added for the event.

7. Click OK to close the [objectname] Behavior window.

Results

You have added a Launch LOB event action.

When the event action defined occurs, query cache in the target component is invalidated and the valid results will display in the component the next time it is refreshed.

When the visual dashboard runs you can open the LOB data by applying the specified event to the object that handles this action. In DataQuant for Workstation, the data opens in the associated system application. In DataQuant for WebSphere, the data opens in the web browser.

Working with modal dialogs in visual dashboards:

You can design templates for modal dialogs and add them to your visual dashboards by using the Show dialog and Close dialog event actions.

About this task

To design a modal dialog and add it to a visual dashboard you need to perform the following actions:

Procedure

1. Create a dialog template to specify visual parameters of the dialog. You can also select a template from the existing ones or edit it.
2. Create a scene, which you want to use for a dialog.
3. In the scene properties, specify the dialog parameters and a dialog template.
4. Add controls, local variables, and event handlers to the dialog scene.
5. Add the Show dialog action to the main scene of the visual dashboard and specify its parameters and behavior.
6. Add the Close dialog action, and specify its parameters.

What to do next

See the following topics for more details about adding modal dialogs to visual dashboards:

Creating dialog templates:

You must create a dialog template or select an existing one for a modal dialog that you want to add to your visual dashboard.
About this task

To create a dialog template:

Procedure

1. In the Project Explorer view, navigate to the Globals node of the visual dashboard to that you want to add a modal dialog.
2. Expand the Globals node to navigate to Dialog Templates. This node contains all available templates for the visual dashboard. You can select a pre-set one or create your own one:
   a. Right-click the Dialog Templates node and select Insert Dialog Template to create a dialog template.
   b. In the Dialog Templates node, right-click an existing dialog template and select Edit Dialog Template to modify an existing template.
3. In the Name field, enter the name that you want to use for this dialog template.
4. Specify the available properties for the template:
   - Content paddings
   - Background
   - Border
   - Shadow
   - Scene Background
   - Title Background
   - Title Font
   - Title Frame
   - Nine Patch
5. When you specified the properties, click Finish to close the window and save the dialog template.

What to do next

See the following topics for more details about specifying the properties of a dialog template:

Specifying background properties for dialog templates:

You can specify background properties for the content of a dialog template to customize its view.

About this task

To specify the background of the dialog template:

Procedure

1. In the New Dialog Template Wizard, select Background from the Properties list.
2. From the Pattern list, select the type of the pattern that you want to apply to the background.
3. In the Color field, specify the color for the background. Use the ellipsis (...) button to select a color from the palette.
4. Move the **Transparency** slider to set the transparency for the background color.

5. In the **Contrast color** field, specify the contrast color for the content borders. Use the ellipsis (…) button to select a color from the palette.

6. Move the **Transparency** slider to set the transparency for the background contrast color.

7. If you selected the gradient pattern, in the **Gradient type** list, select that gradient that you want to apply to the content.

8. If you want to set an image as a background, in the **Image source** field, click the ellipsis (…) button and specify the image that you want to use as a background.

9. If you use an image as a background, you can specify its location in the **Image anchor** list and tiling in the **Image tiling** list.

10. Click **Finish** to save the changes for this dialog template and close the dialog template window.

**Specifying border properties for dialog templates:**

You can specify border properties for a data template to customize its view.

**About this task**

To specify border properties:

**Procedure**

1. In the New Dialog Template Wizard, select **Border** from the **Properties** list.

2. In the **Color** field, specify the color for the window borders. Use the ellipsis (…) button to select a color from the palette.

3. Move the **Transparency** slider to set the transparency for the border color.

4. Move the **Width** slider or enter a particular value to specify the width of the border.

   **Tip:** You can customize the units of measure for borders by selecting a unit from the list of available ones.

5. In the **Top Border Radius** area, you can specify the fillet for the top border of the window. Move the **Top** left slider to set the fillet that you want to apply.

6. If you want to specify the fillet for the top right border separately, select the **Set top right border radius** check box and move the **Top right** slider.

7. In the **Bottom Border Radius** area, you can specify the fillet for the bottom border of the window. Move the **Bottom left** slider to set the fillet that you want to apply.

8. If you want to specify the fillet for the lower right border separately, select the **Set bottom right border radius** check box and move the **Bottom right** slider.

9. Click **Finish** to save the changes for this dialog template and close the dialog template window.

**Specifying content paddings for dialog templates:**

You can specify content paddings for a dialog template to customize the offset between the dialog content and the dialog window borders.
About this task

To specify content paddings:

Procedure
1. In the New Dialog Template Wizard, select Content from the Properties list.
2. In the Horizontal Paddings area, move the Left slider to specify the left and the right padding of the content.
3. If you want to specify the right padding separately from the left one, select the Set right padding check box.

Tip: You can customize the units of measure for paddings by selecting a unit from the list of available ones.
4. In the Vertical Paddings area, move the Top slider to specify the top padding of the content.
5. If you want to specify the bottom padding separately from the top one, select the Set bottom padding check box.
6. Click Finish to save the changes for this dialog template and close the dialog template window.

Specifying NinePatch properties for dialog templates:

Use the NinePatch images that contain the service information for correct image scaling and content placing.

About this task

To specify the NinePatch properties:

Procedure
1. In the New Dialog Template Wizard, select Nine Patch from the Properties list.
2. Select the Use NinePatch image check box.
3. In the Title source field, specify a path to the image for the title area.
   a. Click the ellipsis (...) button to open the Picture Source dialog.
   b. In the Picture Source window, select one of the following options:
      • Linked to specify that the image is in a local or network file. Select Linked, and specify the path to the image file.
      • Embedded to specify that the image is embedded in the visual project. In order for this option to be available you must have images available to the visual project for selection. All images that are available are listed in the Images folder that is under the Globals node in Project Explorer. To specify that the image is embedded, select Embedded. Select the image from the list of available images.
4. In the Content source field, specify a path to the image for the content area.
   a. Click the ellipsis (...) button to open the Picture Source dialog.
   b. In the Picture Source window, select one of the following options:
      • Linked to specify that the image is in a local or network file. Select Linked, and specify the path to the image file.
      • Embedded to specify that the image is embedded in the visual project. In order for this option to be available you must have images available to the visual project for selection. All images that are available are listed in the Images folder that is under the Globals node in Project Explorer.
the Images folder that is under the Globals node in Project Explorer. To specify that the image is embedded, select Embedded. Select the image from the list of available images.

5. Click Finish to save the changes for this dialog template and close the dialog template window.

Specifying scene background properties for dialog templates:

You can specify background properties for the scene that hosts the dialog.

About this task

To specify background properties for the dialog scene:

Procedure
1. In the New Dialog Template Wizard, select Scene Background from the Properties list.
2. Follow the steps 2-10 in the Specifying background properties for dialog templates topic.

Specifying shadow properties for dialog templates:

You can use a customized shadow for a dialog template.

About this task

To specify shadow properties for a dialog template:

Procedure
1. In the New Dialog Template Wizard, select Shadow from the Properties list.
2. Select the Add a shadow check box to apply a shadow to a dialog template.
3. Move the Offset slider to specify the offset of the shadow.
4. Move the Angle slider to specify the angle of the shadow.
5. Move the Radius slider to specify the radius of the shadow.
6. In the Color field, specify the color of the window shadow. Use the ellipsis (...) button to select a color from the palette.
7. Move the Transparency slider to set the shadow transparency.
8. Click Finish to save the changes for this dialog template and close the dialog template window.

Specifying title background properties for dialog templates:

You can specify background properties for the dialog template title.

About this task

To specify the title background properties for the dialog scene:

Procedure
1. In the New Dialog Template Wizard, select Title Background from the Properties list.
2. Follow the steps 2-10 in the Specifying background properties for dialog templates topic.
Specifying title font properties for dialog templates:

You can customize fonts in the titles of dialog templates.

About this task

To specify the font properties for a title:

Procedure
1. In the New Dialog Template Wizard, select Title Font from the Properties list.
2. From the Font list, select a font type for the title.
3. Move the Size slider to set the font size.
4. In the Color field, specify the font color for the dialog title.
5. Move the Transparency slider to set the transparency for the title font.
6. Select or clear the Bold, Italic, or Underline check box depending on the font type that you want to use.
7. Click Finish to save the changes for this dialog template and close the dialog template window.

Specifying title frame properties for dialog templates:

You can customize the title frame in dialog templates.

About this task

To specify the title frame properties for a title:

Procedure
1. In the New Dialog Template Wizard, select Title Frame from the Properties list.
2. In the Horizontal Margins area, specify move the Left slider to specify the left and the right margins of the title.
3. If you want to specify the right margin separately from the left one, select the Set right margin check box.
   
   Tip: You can customize the units of measure for margins by selecting a unit from the list of available ones.
4. In the Vertical Margins area, move the Top slider to specify the top and bottom margin of the title.
5. If you want to specify the bottom margin separately from the left one, select the Set bottom margin check box.
6. From the Horizontal Alignment list, select the alignment for the title.
7. Move the Horizontal shift slider to specify the title frame shift.
8. Click Finish to save the changes for this dialog template and close the dialog template window.

Adding the Show dialog event action:

Use the Show dialog event action to add a modal dialog window that is based on a previously created scene to your visual dashboard.
Before you begin

You must have a previously created scene, which you want to add to the event action, and a dialog template, which is used as a container for the scene.

About this task

To add and define the Show dialog event action, perform the following steps:

Procedure

1. In the editor window, select the object to which you want to add the action.
2. Double-click the event that you want to trigger the action from the Events view. The [objectname] Behavior window opens.
3. Click the Add button.
4. Select Show Dialog in the actions tree and click Next.

   Note: To quickly find the required event action, type the name of the action in the filter text field.

5. In the Add New Action window, specify the action parameters.

6. If you want to open a scene from another visual project, select the Another visual project check box and specify a visual project.
   a. Select the name of the scene that you want to add to the dialog window from the Dialog scene list.
   b. If you have local parameters in the scene that you want to add from the current visual dashboard, you can modify their initial values for the destination scene when event occurs in the Set values for the existing local parameters list.

7. If you want to use a scene from the current project, perform the following steps:
   a. Make sure that the Another Visual Project check box is clear.
   b. Select the name of the scene that you want to add to the dialog window from the Dialog scene list.
   c. If you have local parameters in the scene that you want to add from the current visual dashboard, you can modify their initial values for the destination scene when event occurs in the Set values for the existing local parameters list.

8. In the Title field, specify a title that you want to add to the dialog.

   Note: The Title field value equals the Title parameter value of the scene that you use in the dialog.

9. In the Dialog Location area, specify the location of the dialog window. The available options are:
   • Center location to set the center of the scene as the location of the dialog.
   • Custom location to specify the exact coordinates for the dialog window.

10. Click Next. The next page for specifying Show dialog action parameters opens.

11. Select the Use Close Dialog action result check box to use the parameter that is got in the Close dialog action.

12. If you did not previously specify a parameter for storing the result value of the event action, select either Global parameters or Local parameters from the Result parameter list and click the Add New Parameter button. The Insert
Parameter window opens. In this window, specify a parameter and click Finish. The Insert Parameter window closes and control returns to the Add New Action window.

**Note:** You can add the parameters of the following types: Boolean, Integer, Double, Text, and Literal. Use the added parameter to specify conditions for the event actions execution. For example, if you use a Boolean parameter in the DialogClose event you can create two different actions that are run depending on the True or False returned value.

13. Specify the actions that you want to run when the dialog window closes. For the list of available actions, see the Setting up event action navigation topic.

14. Click the **Add** button. The Add New Action window opens. Specify the parameters for the selected action, and add it.

**Note:** The local parameters that are specified for the dialog scene are available in the Event Parameters category of the Expression Designer. If you have two parameters with the same name but with different values in the main and dialog scenes, the value that is specified for the dialog scene has higher priority and is used as the dialog scene parameter.

15. In the Actions list, you can modify parameters for the added actions.

16. Select the parameter that you want to use from the Result parameter lists and click **OK**. The Add New Action window closes.

17. Click **Finish**. The Show dialog event action is added. The Add New Action window closes.

18. From the [objectname] Behavior window, click **Apply** to apply all actions that were added for the event. Click **OK** to close the [objectname] Behavior window.

**Results**

You created a modal dialog window and added the Show dialog event action to the object.

Adding a Close dialog event action:

Use the Close dialog event action to close a dialog window that is created in the Show dialog event action.

**Before you begin**

You must have a previously created modal dialog window.

**About this task**

To add and define the Close dialog event action, perform the following steps:

**Procedure**

1. In the editor window, select the object to which you want to add the action.
2. Double-click the event that you want to trigger the action from the Events view. The [objectname] Behavior window opens.
3. Click the **Add** button. The Add New Action window opens.
4. Select Close Dialog in the actions tree and click **Next**.
Note: To quickly find the required event action, type the name of the action in the filter text field.

5. In the Add New Action window, specify the return action parameter in the Return value field. You can use the specified value in the event actions that you add in the Show dialog event action.


7. From the [objectname] Behavior window, click Apply to apply all actions that were added for the event. Click OK to close the [objectname] Behavior window.

Results

You added the Close dialog event action to the object.

Adding an Open URL event action:

Use the Open URL event action to allow users to open hyperlinks from visual dashboards.

About this task

For example, you can use label with incorporated Open URL event action, by clicking on which users can go to the company's website. You can also use calculated expressions to create dynamic links that are formed on the basis of user's actions and the context in which they are used.

To incorporate an Open URL event action in an object:

Procedure

1. From the Visual Designer perspective, select the object in the editor window and double-click the event that you want to trigger the Open URL action.

2. In the [objectname] Behavior window, click the Add New Action.

3. Select Open URL in the actions tree and click Next.

   Note: To quickly find the required event action, type the name of the action in the filter text field.

4. In the URL parameters area, specify whether to open the link in a new window or in a new tab.

   Note: The Target parameter is actual only for DataQuant for WebSphere.

5. In the URL field, specify the link or calculated expression. To open the Expression Designer window, double-click in the URL field.

   Note: You can specify either an absolute or relative URL. The relative URL starts with a slash (/) and contains the part of a link that succeeds the base URL. In this case, the absolute URL will be formed by concatenating the base URL (http://host:port/application_path) and relative URL.

6. Click Finish. The Print URL action is added to the list of actions and the Add New Action window closes.

7. In the [objectname] Behavior window, click Apply to apply all actions that have been added for the event.

8. Click OK to close the [objectname] Behavior window.
Adding a Save Cookie event action:

Use the **Save Cookie** event action to store the required data values under specific keys in the application storage for the current user.

**About this task**

In this event action, a user is an operating system user who works with the application or a user who runs the application in a web browser. Each user has a particular storage to keep the created key values. So the key values created by different users do not overwrite each other, even if the keys have the same name.

To add a **Save Cookie** event action to an object, perform the following steps:

**Procedure**

1. From the Events view, select the object in the editor window and double-click the event that you want to trigger. The [objectname] Behavior window opens.
2. Click the **Add New Action** icon. The Add New Action window opens.
3. Select **Save Cookie** in the actions tree and click **Next** to open the Save Cookie action parameters window.

   **Note:** To quickly find the required event action, type the name of the action in the filter text field.
4. Specify the name of the key in the **Key** field.
5. Specify the value of the key in the **Value** field.
6. Click **Finish** to close the Save Cookie action parameters window and return to the [objectname] Behavior window.
7. Click **Apply** to apply all actions that have been added for the event.
8. Click **OK** to close the [objectname] Behavior window.

**Results**

The value of the key is stored in the application storage for the current user and can be retrieved by the name of the key any time.

Adding a Load Cookie event action:

Use the **Load Cookie** event action to retrieve data previously saved in the **Save Cookie** event action from the application storage for the current user.

**About this task**

To add a **Load Cookie** event action to an object, perform the following steps:

**Procedure**

1. From the Events view, select the object in the editor window and double-click the event that you want to trigger. The [objectname] Behavior window opens.
2. Click the **Add New Action** icon. The Add New Action window opens.
3. Select **Load Cookie** in the actions tree and click **Next** to open the Load Cookie action parameters window.

   **Note:** To quickly find the required event action, type the name of the action in the filter text field.
4. Type the name of the key which value you want to retrieve.
5. In the Result option area, click Add New Parameter to create a parameter for saving the value of the key.

   Note: You can save the value of the key to the existing parameter by selecting parameter type and parameter name from the Result parameter lists.
6. Click Finish to close the Load from Session action parameters window and return to the [objectname] Behavior window.
7. Click Apply to apply all actions that have been added for the event.
8. Click OK to close the [objectname] Behavior window.

Results

The value of the key is loaded to the parameter by the name of the key from the application storage for the current user.

Adding a Refresh geolocation event action:

Use the Refresh geolocation event action to update the value of the Geolocation global parameter with the current geolocation data.

About this task

Important: The action is not supported in DataQuant for Workstation.

To add a Refresh geolocation event action, perform the following steps:

Procedure
1. Select the object in the editor window, and in the Events view, double-click the event that you want to trigger the Refresh geolocation event action. The [objectname] Behavior window opens.
2. Click the Add New Action icon. The Add New Action window opens.
3. From the Device tree node, select Refresh geolocation.
5. Click OK to close the [objectname] Behavior window.

Results

You added a Refresh geolocation event action.

Displaying data on a map according to address information retrieved from a query

You can display data from a query result set on a map in your visual dashboards.

About this task

You use calculated columns and custom functions for calculated columns to convert addresses to coordinates in order to use them in your visual dashboards.

You must have created the functions.js file in the DataQuant for Workstation and DataQuant for WebSphere user home directory.

To display data on a map:
Procedure

1. Add the following function to the functions.js file:

```javascript
/**
 * Returns address' geo coordinates
 * @param address address string
 * @category geo
 * @return lan/lng
 */
function getLocation(address) {
    var request = new RSBIHTTPRequest();
    request.open('GET', 'http://maps.googleapis.com/maps/api/geocode/json?address=' + address + '&sensor=false', false);
    var response = '';
    request.onreadystatechange = function() {
        if(request.readyState === 4) {
            response = request.responseText;
        }
    }
    request.send(null);
    var x = JSON.parse(response);
    return x.results[0].geometry.location;
}
```

2. Create a visual query based on the required data table.

3. Add the Coords calculated column to the visual query with the following expression:

```
var loc = getLocation(@[address_column]);
'(' + loc.lat + ',' + loc.lng + ')
```

4. Create a procedure with the following text (replace the values in square brackets with the valid for your case):

```
RUN QUERY "[Path to the visual query]"
SAVE DATA AS [Table name] (ACTION=REPLACE, CONFIRM=NO)
```

5. Create new scheduled task that will run the saved procedure. Data will be updated on the defined scheduled task.

6. Create a visual dashboard displaying the data from the table on a map.

Protecting visual project data

You can compile read-only versions of visual reports and dashboards so that other developers can use them, but they cannot access and modify the underlying data objects.

About this task

To protect visual report and dashboard data:

Procedure

1. Open the previously saved visual report or dashboard which you want to compile in the design mode.

2. In the Project Explorer view right-click the project and select Compile and Save Results as.

3. From the Save [Object] wizard, select whether to save the visual project to a file or repository and click Next.
4. Provide the necessary information in the Save to File or Save to Repository wizard and click **Finish**.

**Results**

You have compiled a read-only version of your visual project. The data associated with the visual report or dashboard is protected and cannot be modified by other users.

**Data source connections and visual projects**

Visual projects (visual reports and dashboards) are intended to be shared across the enterprise. In order to facilitate sharing and the distribution of these visual projects, DataQuant for Workstation ties the query to a connection information alias (rather than associating the object to a specific data source).

You must set up a connection information alias for each data source that will provide data for these objects.

**Specifying data source connections for visual projects**

You set up a connection information alias for each data source that will provide data for the visual project (visual report or dashboard).

**About this task**

When you add the queries that will be used in a visual project, you must specify what connection information alias will be associated with each query.

To change the data source that will be used for a query, you edit the query information and select a connection information alias entry that points to the data source that you want to use. This feature allows users to easily share visual reports and use their own data sources. In addition, it facilitates moving between test and production data sources.

To specify a connection information alias for a data source, perform the following steps:

**Procedure**

1. Open the Insert Connection window.
   
   You can open the Insert Connection window in one of the following ways:
   
   - From the **Project Explorer** view, right-click the **Connections** node for the specific visual project. Select **Insert Connection** from the pop-up menu.
   - From the **Project Explorer** view, double-click the **Connections** node. The Insert Connection window opens.
   - Click a query from the **Workspaces** or **Repository Explorer** view. With the mouse button pressed, drag the query to the **Queries** node in the **Project Explorer** view. A new data source connection is automatically added to the Connections folder of the **Project Explorer** view.

2. From the list of available data sources, select the data source that will be associated with this connection information alias.

3. Specify a unique name for this connection information alias in the **Connection name** field.

4. Click **Finish**. The **Project Explorer** view lists the new connection information alias under the **Connections** node in the tree.
Note: You can also set up a connection information alias for a data source by dragging the data source from the Workspaces view to the Connections folder for the visual project in the Project Explorer view. The connection information alias is added with the name ConnectionN. You can edit the name of the connection information alias by right-clicking the entry and selecting Rename.

Results

You have set up the data source connection information for the visual data object.

Queries and visual projects

Visual projects (visual reports and dashboards) can include one or more queries to run to obtain the result data and display it in the project.

All queries that you use in the visual project must be defined in the Queries folder for that visual project.

You can add queries at any time while designing your project. For each query that you include in the visual project, you must specify what data source will be used. You do that by associating the query with a connection information alias.

Specifying queries for visual projects

Visual projects (visual reports or dashboards) can include one or more queries to run to obtain the result data and display it in the visual project.

About this task

You can create a new query that will be included in the visual project or you can use existing queries.

To specify a query for a visual project, perform the following steps:

Procedure

1. From the Project Explorer view, right-click the Queries node and select Insert Query from the menu.
2. Specify a unique name for the query in the Query name field.
3. Identify the query and specify where it resides in one of the following ways:
   a. If you are creating a new query, select Create a new query.
   b. If you are creating a new visual query, select Create a new visual query.
   c. If you are creating a new analytical query, click Create a new analytical query.
   d. If you are creating a new Cloudant query, click Create a new cloudant query.
   e. If you are creating a new forecast, click Create a new forecast.
   f. If the query exists, select Attach to an existing query. The Select query list appears. Specify the name of the query and where it resides in the Select Query field. You can click Browse to search for the query.
   g. If you would like to insert a static query, select From file.
      - Select Insert Data to copy the query directly into the Queries folder. This option provides greater portability, because the query will stay with the visual project if it is moved from one system to another.
• Select **Use link to file** to create a link to the query’s system directory location. This option limits portability, because if the visual project is moved from one system to another, the query will no longer be available.

• Specify the query to be added in the **Select data file** list. You can click **Browse** to search for the query.

**Note:** You can open DBF, IXF, XML, and TAB files and use them as queries. You can edit the imported result set by filtering, adding calculated columns, applying grouping and aggregation. When you save the visual project, these files can be converted to dynamarts and saved with the visual project. All modifications will be preserved.

4. If you insert a visual query or Cloudant query, you must select a data source connection that you want to use for the query from the **Connection** list. If the **Connection** list is empty, add a data source connection to a visual project.

5. Click **Add Connection** to open the Insert Connection window.

6. From the data sources tree, select the data source that you want to add for the visual project and click **Finish**. The data source connection that you selected is added to the visual project and displayed in the **Connection** list.

7. If you are inserting an existing forecast or analytical query into a visual project, you can edit connection settings for subordinate queries in the Edit Connections window.

8. If you are inserting a new forecast or analytical query you cannot set connections for subordinate queries because these queries are not specified yet.

9. Click **Finish**. The Project Explorer view lists the query under the **Queries** node in the tree. If the query requires a parameter, a **Parameters** folder is added. You will automatically save the query when you save the visual project.

**Results**

**Note:** You can also add a query by dragging the query from the Workspaces view to the **Queries** node of the visual project in the view. The selected query is added to the **Queries** folder. A new data source connection is automatically added to the **Connections** folder of the Project Explorer view.

---

**Adding custom JavaScript functions to visual projects**

You can use your own JavaScript functions in visual reports and dashboards with the help of JavaScript Modules.

**About this task**

You use the JavaScript Module editor with syntax highlighting to write and edit your own functions. To open the editor, select **File > New > Other**. In the New wizard, select **DataQuant > JavaScript Module**.

After you have created a JavaScript Module, you can save it in a repository or file system, and then add it to a visual report or dashboard.

To add custom JavaScript functions to a visual project:

**Procedure**

1. Open the Insert JavaScript Modules window. In the **Project Explorer** view, right-click the **Globals** node, and then select **New > Insert JavaScript Module** option.
2. In the Insert JavaScript Modules window, decide whether you need to create a JavaScript Module or attach an existing one.

3. If you decide to create a JavaScript Module, specify the required name in the JavaScript Module name field, select Create new, and then click Finish. The empty JavaScript Module opens in the separate editor window. Write the required JavaScript functions and close the editor.

4. If you decide to attach an existing JavaScript Module, select Attach to an existing, and then specify the required module form a repository or file system. Click Finish.

Note: You can view comments added in the JSDoc format to the script. These comments are visible to a user in the Expression Designer.

5. You can view the attached JavaScript Modules in the Project Explorer view under the Globals > JavaScript Modules node.

Results

All the functions defined in the attached JavaScript Module can be found in the Expression Designer under the User node. You can use them as other expressions in your visual project.

Creating JavaScript Modules

You create JavaScript Modules to customize JavaScript functions that you want to use in your visual projects.

About this task

To create a JavaScript Module, do the following actions:

Procedure

1. From the main menu, select File > New > Other to open the New wizard.
2. In the New wizard, expand DataQuant Objects folder and select JavaScript Module.
3. Click Next to open the New JavaScript Module wizard.
4. Specify the name for the JavaScript Module in the Name field and click Finish to open the JavaScript Module editor.
5. Enter JavaScript functions in the editor.
6. Start each function with the keyword 'function'.
7. Add a comment before each function.
   The application interprets this comment as the description of the function.
8. Optional: Define a category to which you add a new function. Use the '@category' tag in the comments to add it. If there is no such category, it will be created.

Note: The following categories already exist in the application:
- Columns
- Aggregation
- Conversion
- Date and Time
- Information
- Logical
9. To verify the correctness of the JavaScript functions that you entered in the editor, click the Check JavaScript syntax and structure toolbar button.

10. After you entered the required JavaScript functions, save the JavaScript Module to a repository or a file.

---

**Working with the Visual Designer**

In DataQuant for Workstation, the Visual Designer perspective presents an editor window, several views and the menus that you will use to create visual projects.

**About this task**

You will use the following when you create either type of visual project:

- **Project Explorer**
  This view displays in a tree structure all the elements and objects that will be included in the visual project.

- **Editor**
  The editor window displays a canvas on which create your visual project. From the editor window, you can switch between different modes that allow you to design and check the final view of your visual project.

- **Properties view**
  This view displays all the properties of each element and object that is included in the visual project. When you select an element from the Project Explorer, its properties are displayed in the Properties view. When you insert or select an object, its properties are displayed. When you make changes to an element's or object's properties, you change the way the element or object appears or behaves. You can make changes to properties directly in the Properties view.

- **Events view**
  This view displays all the events available for each element and object in the visual project.

  When you select an element from the Project Explorer, the events available for this object are displayed in the Events view. By using the Events view you can assign different event actions to scene objects and thus provide the scene interactivity and navigation at run time.

- **Palette view**
  This view displays all of the graphic objects that you can insert in a visual project. You select objects from the Palette view and insert them in the editor.

  For each object that you insert, an entry is made in the Project Explorer and the properties for the object are displayed in the Properties view.

---

**Working with the Project Explorer**

The Project Explorer view is used when you are working with visual projects. It lists in a tree structure each visual report or visual dashboard that you currently have open.
About this task

As you add elements to your visual projects, each element is added to the Project Explorer tree as a folder. When you insert an object into your visual project, the object is also added to the Project Explorer tree as a folder. Depending on your element or object, additional folders are added.

The Project Explorer works in conjunction with the Visual Designer editor and the Properties view. When you select a visual project element from the Project Explorer all of the design objects that have been defined for the visual project element are displayed in the editor window and the properties for that element or selected object are displayed in the Properties view.

Each type of visual project includes different structural elements. Each visual report always includes the following high level folders:

- Connections
- Globals
- Queries
- Fixed pages
- Main page

Each visual dashboard always includes the following high level folders:

- Connections
- Globals
- Queries
- Scenes

As you add design objects to your visual projects, additional folders representing those design elements are automatically added to the Project Explorer tree.

Exporting images from visual projects

You can export images that are stored in a visual project to a graphic file.

About this task

Visual projects might contain different images that help to demonstrate the required information clearly. You can find these images in the Project Explorer view by expanding the Globals > Images node. If you need, you can export the images to graphic files.

To export an image to a file:

Procedure

1. In the Project Explorer view, select the required visual project.
2. To view the list of images, expand the Globals > Images node.
3. Right-click the required image and select Save at file from the menu.
4. To export all images that are stored in the visual project, right-click the Images node and select Save at File from the menu. The images will be packed in a ZIP archive.

Related tasks:

"Inserting picture objects” on page 327

You use the Picture object to insert an image into your visual project.
**Viewing object references**
You can view all references to an object used in the visual project in the Show References window.

**Before you begin**
You must have a visual project opened in the **Visual Designer** editor.

**About this task**
To view object references:

**Procedure**
1. In the Project Explorer view, right-click the required object and select **Show References** from the pop-up menu. The number of objects that have references to the selected object is displayed in the **Show References** window. If the selected object is referenced by other objects, the **Advanced** button becomes available.
2. To view all references, click **Advanced**, and then navigate to the object references in the tree.
3. You can use the arrows to navigate to the previous or to the following object that references the selected one.
4. To expand or collapse the tree of objects, click **Expand** or **Collapse**.
5. To view the selected referenced object in the **Project Explorer** view, click **Show in Project Explorer**.

**Working with the editor**
The **Visual Designer** editor window displays a canvas on which you create your visual project.

**About this task**
From the editor window you can switch between the modes that allow you to design your visual project and check how the project is displayed for users in different rendering modes.

You can open multiple visual project elements at the same time in the **Visual Designer** editor window. Each opened element appears as a separate canvas with an identifying tab along the top of the editor window. When you click the canvas tab in the editor window, that canvas becomes active. In order to make changes to a specific visual project element, that element must be active in the editor window. You can also activate a canvas by selecting the element of the visual project in the **Project Explorer** view.

The tabs at the bottom of the window allow you to toggle between the following modes:

- **Canvas mode**
  You use **Canvas** mode to add content to your visual project. When you add objects they are represented in the editor by placeholder objects.

- **Design mode**
  You use **Design** mode to arrange and modify objects, which you added in **Canvas** mode, to adjust the content of your visual project to different screen resolutions.
• **Runtime mode**
  You use Runtime mode to test the content in your visual dashboard. In Runtime mode, graphic objects are drawn as they will appear to your users, queries are executed, object events are generated, and associated actions are performed. In addition, navigation features are executed. The Runtime mode of the editor displays your visual dashboard as the end-user will see it.

• **Preview mode**
  You use Preview mode to test the content in your visual report. In Preview mode, graphic objects are drawn as they will appear to your users, queries are executed, object events are generated, and associated actions are performed. The Preview mode of the editor will display your visual report as a user will see it.

You can control how your visual project will be rendered in the Runtime and Preview modes by using the rendering mode list in the workstation menu bar.

Depending on the type of the visual project the following rendering mode options can be available:
- **HTML** optimizes the content to be viewed in a web browser.
- **HTML5** optimizes the content for better performance in modern web browsers and on mobile devices.
- **Flash** optimizes the content to be viewed in a Flash player.
- **PDF** optimizes the content to be printed.

**Editor controls in the visual designer**

In the visual designer the editor rulers, guides, and grids help you to position visual objects precisely.

**Rulers**

When visible, rulers appear along the top and left side of the available window. The ruler origin also determines the grid’s point of origin.

To show or hide rulers, select **View > Rulers**.

Pixels are used as the default unit of measurement. You can change the default unit of measurement on the Visual Designer page of the Preferences window.

To change the unit of measurement:
1. Select **View > Preferences**
2. Select **Visual Designer** from the tree.
3. Select one of **Visual Designer Ruler Unit** to specify the default units for the ruler. Your options are pixels, inches, or centimeters.

**Guides**

Guides appear as nonprinting lines above the scene and the objects it hosts.

To create a vertical guide, click somewhere on the horizontal ruler.

To create a horizontal guide, click somewhere on the vertical ruler. A new guide is drawn over the whole scene at the chosen position. You can also move the guides you created to new positions using the drag-and-drop method.
To remove a guide, select the guide and with the left mouse button pressed, drag it outside the ruler.

**Grid**

Use the grid for laying out elements symmetrically.

The grid is also drawn as nonprinting. By default, the grid displays whenever you open the **Visual Designer**.

To show or hide the grid, select **View > Grid**.

The grid's point of origin is located at the coordinates 0,0, which normally is at the center of the scene.

**Note:** The changes applied to the ruler's unit of measurement do not have any affect on the grid. Grid units always have a fixed size.

**Snap to Grid**

Use the **Snap to Grid** option to unbind objects and control their moving regardless to the visibility of the grid.

To move the scene components regardless to the visibility of the grid, select **View > Snap to Grid**. The objects are unbound and can be moved on the scene regardless to the grid.

**Working with the Properties view**

Each element and object that is included in a visual project has unique properties that dictate how the element or object will appear in the finished visual project. Properties for each element or object in a visual project are displayed in the **Properties** view.

**About this task**

Properties vary depending on the visual project element or object. They can include such things as location, value, size, alignment, line style, transparency, and color. They can also include properties that define how an object's appearance changes depending on a user's action.

**Specifying object properties**

Properties and their corresponding values for each element or object in a visual project are displayed in the **Properties** view.

**About this task**

The properties presented in the **Properties** view vary depending on the element or object that is selected in the editor window or in the **Project Explorer** view. Properties are listed with their current or default values.

There are many ways for you to modify the property values. You can modify the property values using constants, mathematical expressions, function expressions, or property expressions. For many properties, you can select a value from the property's list box of accepted values. Property values that you specify must match the data type required for the property.
To specify or modify individual object properties from the Properties view:

Procedure
1. The Properties view opens automatically when you are in the Visual Designer perspective. If the Properties view is not open, you can open it by selecting Window > Show View > Properties. To open the Visual Designer perspective, select Window > Open perspective > Visual Designer.
2. Select from the Project Explorer view, or the editor window, the element or object whose properties you want to view or modify. Once selected, the properties for that element or object are displayed in the Properties view.
3. Click the property cell for the property that you want to modify and enter the value. Depending on the property, the values that you enter could be a constant, a global or local parameter, a column name that is associated with the results of a query, a selection from a list of available values or an expression. The value you enter must match the data type required for the property.
4. You can use the Expression Designer to help you specify property values. Click the Edit with Expression Designer toolbar button in the Properties view to open the Expression Designer.
5. You can use the Variables bar to help you specify property values that will use global or local parameters or column names from a set of query results.

Specifying properties for groups of objects
You can specify properties for groups of objects that you have selected in the editor window.

About this task
To specify property values for a group of objects:

Procedure
1. In the Editor window, click the visual project canvas that is active.
2. With the left mouse button pressed, move your mouse to enclose the objects whose properties you would like to modify.
3. The common properties of all the objects that you selected are listed in the Properties view. If the values of a displayed property are the same for all selected objects, it will be displayed in the Value column. If any of the selected objects has a different value, the field is blank.
4. Click the property field for the property that you want to modify and enter the value. All objects selected will be modified.

Using the Variables bar to specify property values
You can use the Variables bar to specify a property value. The Variables bar is located in the menu bar.

About this task
From the Variables bar, select one of the following options to set the object’s property value:
• a global or local parameter that has been defined for the visual project.
• a column name from a set of query results.

To specify a property value using the Variables bar:
Procedure

1. In the Variables bar, select one of the following items from the first drop-down list:
   - Global parameters populates the second drop-down list with all the global variables that have been defined. Select the global variable that you want to use.
   - Local parameters populates the second drop-down list with all the local variables that have been defined. Select the local variable that you want to use.
   - One of the queries that has been defined for your project. The names for each of the columns that are included in the query results are listed in the second drop-down list. Select the column name that you want to use.

2. Click the Link Variable to Object icon. A red dot appears.

3. Pass your mouse over the red dot, and drag the dotted line that appears from the red dot to a property cell that is listed in the Properties view.

4. Click the property field. The column or parameter name is inserted as the value for the property that you selected.

Using the Expression Designer to specify a property value

The Expression Designer can be used to specify any property value that is associated with a selected object.

About this task

The Expression Designer is an alternative way of specifying an object's properties. Rather than type an object's properties directly in the property cell of the Properties view, you use the fields and buttons in the Expression Designer to help you create and format the property value. There are advantages to using the Expression Designer to specify your object's properties. For each property that can be specified for an object, you are given templates that can help you format the value of that property. In addition, you have more space available to work and view your property value (which can grow quite large if expressions are used).

To specify a property value using the Expression Designer:

Procedure

1. Select a property from the Property view. Click the Edit with Expression Designer toolbar button in the Property view. The Expression Designer window opens.

2. In the left pane of the window, each element of your visual project is listed in the Expression Designer tree. Expand each element to see a list of properties for the element.

3. Each of the built-in functions that can be included in property value expressions are listed in the right pane of the window. Expand the function categories to see what functions are available. You can filter the list of functions that are displayed using the filter field. You can type one or more letters in the filter field, and all the functions that start with the entered letter (or letters) are listed. You can also type in a specific function name and it will be listed.

4. The property that is being edited is displayed above the property value field. You can use both properties and functions to build your property value in the property value field. Double-click a function to insert it into the property value field. Double-click a property to insert a reference to it in the property value field. In both cases, the function or property is appended to what is already in
the property value field. To replace content, highlight what you want to replace in the property field and then double-click the property or function. To insert content, position the cursor in the property field and then double-click the property or function. Functions are placed in the property field in template form. Depending on your logic, you will replace the parameters with your own values.

5. The following buttons, located below the property value box, provide additional help when creating a property expression:
   - **Undo**: Select this button to undo the last edit applied to the expression that you are building in the property value field.
   - **Redo**: Select this button to redo the last edit applied to the expression that you are building in the property value field.
   - **Edit**: Use this button to build an expression that will define a new property value. The property for which you are building an expression is displayed above the property value field. To change the property under edit, select the new property from the **Expression Designer** tree in the left pane of the window, and click the **Edit** button. The new property that will be edited is displayed above the property value field. You can set or edit multiple property values using the **Edit** button without closing the Expression Designer window. Each property that you modify is marked in the **Expression Designer** tree with an asterisk (*).
   - **Open FormatNumber dialog**: Select this button to open a list of available templates that you can use to format a number.
   - **Simplify expression**: Click this button to parse the expression that you have built in the property value field. The expression will be reviewed and simplified where possible.
   - **Select color**: Select this button to open the Color window. From the Color window you will select a color and click **OK**. The hexadecimal color number will be substituted for the highlighted color parameter displayed in the property value field.

6. Click **Apply**. The property value is updated.

7. Click **OK**. The property value is updated and displayed in the **Properties** view. The Expression Designer window closes.

**Writing property expressions**

Property expressions are conditional or mathematical expressions that you can specify as the value for an object's property. These expressions are evaluated to obtain the property value at runtime when a dashboard or report is displayed.

Property expressions enable you to create dynamic values for your object's properties. Many business decisions depend on factors such as market conditions, sales, operational expenses, new locations, profit and loss margins, and customer satisfaction. These factors are subject to change.

**Elements of a property expression:**

Property expressions are composed of one or more elements that will supply values at runtime.

The following elements make up property expressions:

- **Functions**: Examples of functions include mathematical or statistical operations such as Min() and StdDev(). You can use functions to calculate values, convert values, or to specify Boolean conditions.
- **Operators**: Examples of operators include =, >, and <. You can use operators to perform arithmetic, to compare values, or to concatenate values.

- **Column Names**: Examples of column names might be any name that has been assigned to a column of data returned by a query. You can use these column names in your expressions.

- **Formatting symbols**: Examples of formatting symbols are $, #, and %. You can use these symbols to format numeric and date/time values.

- **Constants**: Examples of constants are values for any valid data type, such as True and False for a Boolean data type. Use constants in functions, with operators, with column names, and by themselves to specify a property value of the appropriate data type.

**Required data types in return property values:**

When you specify an expression for a property value, the expression must return a value of the data type required by the property.

Object properties and their supported data types are listed by object in the Object Properties Reference. To return the value that you want in the data type required, you can convert it to the appropriate type using one of the built-in conversion functions.

**Syntax rules for writing property expressions:**

You must use the correct syntax when you write property expressions.

The following table describes some basic rules for typing property expressions:

<table>
<thead>
<tr>
<th>Task</th>
<th>Rule</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>Display a calculated value, such as a parameter or column name</td>
<td>Use equals sign (=) for calculated expressions</td>
<td>=cust_num</td>
</tr>
<tr>
<td>Display a literal value</td>
<td>Type the literal exactly as you want it to appear in text</td>
<td>Sales Organization</td>
</tr>
<tr>
<td>Display a text string in an expression</td>
<td>If the string is part of a combined expression that also includes calculated values, columns or parameters, it must be surrounded by quotation marks (&quot;)</td>
<td>=&quot;My name is: &quot; + fname</td>
</tr>
<tr>
<td>Concatenate two or more text strings</td>
<td>Use the plus sign (+) to concatenate text strings</td>
<td>=Click here to go to&quot;+Dept+&quot; department</td>
</tr>
<tr>
<td>Add two or more numeric values</td>
<td>Use the plus sign (+) to add constants or parameters of numeric data</td>
<td>=Param1 + 5</td>
</tr>
<tr>
<td>Insert a carriage return in an expression</td>
<td>Type newline. If the expression elements are the same data type, use a plus sign (+) before and after. Otherwise, include newline in the Concat() function</td>
<td>=city+newline+state</td>
</tr>
</tbody>
</table>
### Calculated expressions versus literals:

All property expressions start with an equals sign (=). This indicates that the property must be computed rather than interpreted as a fixed series of characters.

The following examples show two property expressions for the `ToolTipText` property of a `Button` object:

**Example 1**

This example defines a literal value: `2+3`

The tooltip will literally display the characters: `2+3`

**Example 2**

This example defines a calculated value: `=2+3`

The tooltip will display: `5`

### Examples of property expressions:

Some property expression syntax rules are best described using examples.

**Writing an if() statement**

You can use an `if()` function to create a Boolean (true/false) property value. For example, if you want to display the object as blue if the condition specified is met and yellow if it is not met, you might write the following expression:

```
Color = if(S_terrain="water", Blue, Yellow)
```

In this example, the condition of the object is based on a parameter, `S_terrain`. The object might be a polygon in a map layout or a data symbol shown on a chart. If the value of the scene parameter is water, the color of the object is blue. Otherwise, the color is yellow.

You can specify different conditions for the `if()` function, such as a column value or a numerical threshold. For example, you can make your visual project be responsive to changes and show their significance in the appearance of the objects in the project. Profit or loss, for example, can be displayed visually using color: black for profit, red for loss. The following property expression can be used to distinguish profit and loss in the color property of the object:

```
Color = if (sales>=expenses, Black, Red)
```

**Displaying column data in a text string**

You can enhance the data points for a layout object by adding a `Label` object that reveals information about the record returned. For example, you might want each (data point) in a tree chart to show the name of a department in a store, preceded by the word Department. You could insert
a **Label** object into the data template for the layout object and then specify a property expression which combines the column name and a literal string.

Value = "Department: \n"+newline+dept_desc

In this expression, dept_desc is a column returned by the query to your data template.

**Working with the Events view**

Events are actions that are built into visual dashboards by designers. These actions provide the dashboard with all the necessary navigation and interactive functionality. The Events view displays these events.

The Events view consists of two panes.

The *upper pane* displays the events tree and the filter text field that you can use to quickly find the required event. The events tree contains all events that are available for the object that you select in a visual project. Each object has a default event that is displayed in the first root node of the tree. All other events that you can assign to the selected object are grouped by categories and displayed under the appropriate nodes of the tree.

The *lower pane* of the Events view includes the list of event actions defined for the selected event and instruments for their editing.

To open the Events view, perform the following steps:

1. From the main menu, select **Window > Show View > Other** and expand **Visual Designer Category**.
2. Select **Events** from the list of options and click **OK**.

When assigning an event to an object, designers can specify additional actions to be triggered when the event occurs. See the [Setting up event action navigation](#) topic for information about how to assign events to objects and how to specify the actions that are triggered when the event occurs.

For information about how to assign events and event actions to visual dashboards, see the [Assigning events and setting up event actions](#) topic.

**Working with the Palette view**

The **Palette** view displays all of the graphic objects that are available for insertion into your visual project.

**About this task**

The objects are grouped into the following *palettes* (also referred to as *drawers*):

**Frequently Used**

The *Frequently Used* palette contains 10 mostly used objects for easier navigation among them. A user can add objects to this palette or remove them in the Customize Palette dialog.

**Note:** To open the Customize Palette dialog, right-click any object on the palette and select **Customize**.
Primitives
The Primitives palette contains basic graphical objects that you can use to display text, rectangles, polygons, pictures and alignment panels.

Controls
The Controls palette contains graphical objects that you can use to display standard edit controls, such as sliders, buttons, and combo and list boxes. In addition, this palette includes the Slicer object that provides an easy way to filter the multidimensional data that is displayed in a layout object and the DateTime object that provides a convenient way for users to enter date and time information. The Controls palette is only available when creating visual dashboards.

Data Symbols
The Data Symbols palette contains graphical objects that are designed for inclusion in a layout object to augment a layout object's ability to represent its query results data. Data symbols can also be used independently of layout objects, and as such they are similar to the objects that are contained in the Primitives palette.

Layouts
The Layouts palette contains graphical objects that you can use to display the data that has been obtained as a result of a query. Layouts include a wide assortment of charts, graphs, maps, tables and grids.

Containers
The Containers palette contains the embedded scene object that you can use to link two scenes together and the embedded content object that you can use to add external content, such as a text file or HTML page, to a visual dashboard.

Miscellaneous
The Miscellaneous palette contains the Timer object, which is used to automate the occurrence of an event based on time intervals.

Connectors
The Connectors palette contains graphical objects that can be used to display a visual link between data points in a layout object or between two graphical objects that have been inserted in a visual report or dashboard. Connectors include an assortment of straight and curved lines.

Custom
The Custom palette contains objects from any of the other palettes. You are responsible for placing the objects that are on the Custom palette. You can place objects you have set custom properties for, or objects that you use frequently on the Custom palette. You place these objects in the custom palette so that they are easily accessible.

The Palette view also contains the tools that you can use when working with visual projects. You can find the following tools at the top of the Palette view:

Select  The Select Tool is used to select an object.

Grab and Pan
The Grab and Pan tools are used to move a dashboard scene around within the Screen editor window.
These tools are available when creating visual dashboards only.

Connectivity
The Connectivity tool provides users with the capability to define
dependencies between dashboard control objects (such as combo or listbox objects) and data-driven layout objects (such as a grid or bar chart object). You can also use the Connectivity tool to define dependencies between non-query driven objects such as text box objects and label objects.

The Connectivity tool is available when creating visual dashboards only.

**Inserting palette objects into a visual project**

From the Palette view, you can insert any graphical object that is listed into your visual project.

**About this task**

All objects from the Palette view can be inserted in a visual dashboard. Not all objects available from the Palette view can be inserted in a visual report. The objects that cannot be inserted in a visual report have interactive properties, and it does not make sense to insert them in visual reports because visual reports are not interactive.

To insert a graphical object into a visual project perform the following procedure:

**Procedure**

1. Open the Palette view.
   
   Generally, the Palette view opens within the editor window when the visual project is active.
   
   If the Palette view is not opened, open it using the following method:
   
   a. Select Window > Open perspective > Visual Designer
   
   b. With the Visual Designer perspective open, select Window > Show View > Other
   
   c. From the Show View window, select General > Palette and click OK
      
      The Palette view opens.
   
2. In the editor window, select the visual report page or visual dashboard scene to which you want to add the object. If the page or scene is not open in the editor, then double-click the page or scene in the Project Explorer.
   
3. There are two ways that you can insert an object into the editor window:
   
   a. From the Palette view, click the object that you want to insert into your visual project, then click in the editor window. A representation of the object is drawn on the canvas for the currently active visual project element that is in the editor window. The object is positioned in a default location and drawn to a default size. The properties of the object are displayed in the Properties view. The name of the object is added to the Project Explorer tree.
   
   b. From the Palette view, click the object in that you want to insert into your visual project, then, click in the editor window, and drag diagonally across a rectangular area to define the location and size of the object. A representation of the object is drawn on the canvas positioned in the location that you have drawn at the size you have drawn. The properties of the object are displayed in the Properties view. The name of the object is added to the Project Explorer tree.
   
4. After inserting your object, you can modify the object’s properties using the Properties view.
   
5. To alter the presentation of the objects as they appear in the Palette view, right-click anywhere in the Palette view to open a pop-up menu. Depending on
how you want to view the objects in the Palette view, select one of the commands that are offered in the pop-up menu.

**Modifying the Palette view display settings**
You can vary the way objects are displayed in the Palette view.

**About this task**
To vary the way objects are displayed in the Palette view:

**Procedure**
1. Open the Palette view.
   Generally, the Palette view opens within the editor window when the visual project is active.
   If the Palette view is not opened, open it using the following method:
   a. Select Window > Open perspective > Visual Designer
   b. With the Visual Designer perspective open, select Window > Show View > Other
   c. From the Show View window, select General > Palette and click OK
      The Palette view opens.

   **Note:** If there is no active visual project, the palette will be unavailable. To make the Palette available, you must first select a visual project. For example, a Visual Report or Visual Dashboard.

2. Each group of objects is displayed in a separate drawer. Clicking on the drawer will alternately expand and collapse the drawer, displaying or hiding the object icons.

3. Right-click anywhere in the Palette view. From the pop-up menu select one of the following options:
   - **Layout → Columns** to arrange the object icons and their names in two columns.
   - **Layout → List** to arrange the object icons and their names in a list.
   - **Layout → Icons Only** to display only the object icons.
   - **Layout → Details** to display the object icons, their names and a description.

4. From the pop-up menu, select **Settings** to open the Palette Settings window where you can set the font that is used to display the name and descriptions of the icons, specify layout options, and specify drawer options:
   - To specify font options, click **Change**. A Font window opens where you can change the default font.
   - To specify layout options, select from the **Layout** radio group. Also, select the **Use large icons** check box to display larger icons.
   - To specify drawer options, Select from the **Drawer options** radio group. Your choices are:
     - Always close when opening another drawer
     - Close automatically when there is not enough room
     - Never close

   **Note:** You can also control the opening and closing of drawers by clicking the pin icon that appears in the drawer of each palette type. Clicking the pin open icon (it becomes highlighted) will cause the current drawer to stay
open when you open another drawer. Clicking a highlighted pin will turn
the pin open icon off and cause each drawer to be closed when another
drawer is opened.

5. From the pop-up menu, select Customize to open the Customize Palette dialog.
In the Customize Palette window, you can change the name and description of
palette objects, rearrange how objects are displayed in the palettes, delete
objects from the Custom palette, and control the visibility of palettes on start
up.

Changing an object’s default property value
Each object that you add to a visual project is inserted and drawn using the
default values that have been set for the object’s properties. You can set new
default properties for these objects so they are drawn and inserted as you most
frequently use them.

About this task
To change an object’s default property value:

Procedure
1. In the Palette view, locate the object that you want to change.
2. Click the object to place it in the available editor window.
3. In the Properties view, change properties.
4. When you are finished changing properties, return to the editor window,
right-click the object and select Set as Default from the pop-up menu. The
object’s default property values are changed. Each time you select this type of
object from the Palette view, it will be inserted and drawn in the editor
window according to the new default properties.
5. To restore an object’s original default properties right-click the object and select
Restore Default Properties from the pop-up menu. If the menu item is
unavailable in the Palette view, the object’s default properties have not been
changed.

Using the Connectivity tool
The Connectivity tool enables users to easily define dependencies between
dashboard control objects (such as Combo or ListBox objects) and data-driven
layout objects (such as a Grid or BarChart object).

About this task
You can also use the Connectivity tool to define dependencies between non-query
driven objects such as TextBox objects and Label objects. The Connectivity tool is
only available when creating visual dashboards.

To connect a source object to a target object using the Connectivity tool:

Procedure
1. Create the source object. Select the object that will be the source object from the
Palette view. Insert the object in your dashboard scene. In most cases the object
that you select is one that can pass meaningful information to the target object.
Typical source objects are control objects such as the ListBox or Combo objects.
2. Create the target object. Select the object that will be the target object from the
Palette view. Insert the object in your dashboard scene. In most cases you will
select a target object that can change in some way after receiving meaningful
information from the source object. Typical target objects are layout objects. When you create a layout object, you will specify the query that will supply the data for the layout object. The query should be placed in the Queries folder for the dashboard.

3. Click the Connectivity tool in the Palette view. Click the source object. A red dot appears. Click the target object.

4. If the target object is not a layout object, the source and target objects will be connected using a property value. The property value that is used is preset and based on the type of objects that are being connected. The Connectivity tool processing is complete. Review the properties for the target object as displayed in the Properties view. You will see the reference to the source object in the selected property cell. You can make manual changes to modify the property used to connect the two objects.

5. If the target object is a layout object, the source and target objects are connected using a parameter. You set up the parameter information in one of the following ways:
   - If the query for the target layout object does not include any parameters, then the Select the column to be connected to the source object page of the Connection Options wizard opens. Select the column from the Columns list that will be used to filter the query result data displayed in the target object based on the information in the parameter passed from your source object. Click Finish. You will be notified that a parameter has been added to the query for the layout object. Click OK. The two objects are connected. The Connectivity tool processing is complete.
   - If the query for the layout object already includes parameters, the Select a connection type page of the wizard opens. Click Set an existing query parameter to use one of the existing query parameters to pass connecting information from the source object to the target object. Click Next. The Select a query parameter page of the wizard opens. Select the parameter from the list of available parameters that have been defined for the query. Click Finish. The Connectivity tool processing is complete.
   - Click Filter by an available table column to select an available table column and create a new parameter. The Select the column to be connected to the source object page of the Connection Options wizard opens. Select the column from the Columns list that will be used to filter the query result data displayed in the target object based on the information in the parameter passed from your source object. Click Finish. You will be notified that a parameter has been added to the query for the layout object. Click OK. The two objects are connected. The Connectivity tool processing is complete.

Using the timer object
You use the Timer object to dictate or automate the occurrence of an event action based on time intervals.

About this task
You can find the Timer object in the Miscellaneous palette.

To insert a Timer object:

Procedure
1. In the editor window, select the visual dashboard scene to which you want to add the Timer object. If the scene is not open in the editor, then double-click the scene in the Project Explorer view.
2. From the Palette view, select the Timer object and click the editor window. The Timer object is drawn on the canvas in the editor window and it is added as a node in the Project Explorer.

3. From the Properties view, set the EventInterval property value. You will specify a time value in this property. The default is set to one (1) second.

4. You can create multiple Timer objects each having a different value for the EventInterval property. For example, one Timer object can have an EventInterval set to one second. Another Timer object can have an EventInterval set to five seconds.

5. In order for the Timer object to have any effect, you must set an event action to occur each time the EventInterval time elapses. For example, if you set the EventInterval property to one second, the event action that you assign to the specific Timer object will occur every one second.

6. To set an event action for the Timer object, select the Timer object from the Project Explorer view.


8. Click the Add New Action icon. The Add New Action wizard opens. You can choose one of the following actions to occur on every tick of the EventInterval:

   - Jump to new location
   - Navigate to embedded scene

   **Note:** The Navigate to embedded scene action is only available to objects other than the EmbeddedScene object if the EmbeddedScene object is in the current scene.
   - Execute SQL statement

   **Note:** The Execute SQL statement action is only available to objects if one or more repository connections have been defined for the visual project in the Connections folder.
   - Execute procedure
   - Set values
   - Navigate back
   - Print scene
   - Execute shell command
   - Show message
   - Execute JavaScript
   - Export to Excel
   - Set focus
   - Reload scene
   - End session
   - Export to PDF
   - Send Email
   - Send Event to Embedded Scene
   - Save Cookie
   - Load Cookie

Although you can select any of the actions, the most common use of the Timer object is to control the behavior of another object. To control the
behavior of another object at each tick of the EventInterval, select the Set values action. If you intend to control the behavior of another object, then you must have created the object before you start the process of assigning an action to the Tick event.

9. Click Next. The next window that opens depends on the action that you selected. If you selected the Set values action, the Set values action parameters window opens.

10. Double-click the Parameter cell. The Expression Designer opens.

11. The Expression Designer tree lists each object that is included in the selected visual dashboard. Expand the node for the object whose behavior will be changed at each tick interval. A list of the object's properties is displayed.

12. Select the property of the object that will change at each tick interval. For example, if the object is a Label object, select the Text property. Specify what the value of the text property will be at each tick of the interval using one or more of the available functions.

13. Click OK. At runtime, with a tick interval of one second, the content of the Label object will change to the new value that you specified every one second.

Adding primitive objects
Primitive objects are graphic objects such as text, lines, shapes, alignment panels, and pictures.

About this task
The following primitive objects are provided:

Label Displays text, with or without a rectangular frame.
Line Displays a straight line segment connecting two points.
Picture
Displays one of these image types:
- Graphics Interchange format (.gif)
- Joint Photographic Experts Group (.jpeg)
- Portable Network Graphics (.png)
Rectangle Displays a rectangle.
Round Rectangle Displays a rectangle with rounded corners. The radius of the curve at the corners is set in the Radius cell in the Properties view.
Ellipse Displays a standard ellipse with foci aligned with the x- or y- axis.
Arrow Displays a straight line segment with an arrowhead on one endpoint.
Double Arrow Displays a straight line segment with an arrowhead on each endpoint.
Polyline Displays a series of line segments connecting vertices. The vertices are specified using a PointSet data type.
MultiPolyline Displays one or more unique polyline objects that are connected or disjointed.
Polygon
Displays a series of line segments connecting vertices. The vertices are specified using a PointSet data type, where the final segment connects the first and last vertices.

MultiPolygon
Displays a series of one or more disjointed polygon objects.

Alignment panel
Groups multiple objects relative to a single point so they all move in unison when based on a single location value.

Inserting text objects:
You can use the Label object to insert text in your visual project.

About this task
To insert text into a visual project:

Procedure
1. In the editor window, select the visual report page or visual dashboard scene to which you want to add the object. If the page or scene is not open in the editor, then double-click the page or scene in the Project Explorer.
2. From the Palette view, select the Label object and insert it in the editor window.
3. From the Properties view, type the text that you want in the Text property cell. There are many additional properties that you can specify that will dictate the appearance of the text or specify how the Label object will behave on the visual report page or visual dashboard scene. There are several ways to specify property values for the Label object. You can select values that are listed in the Properties view, type values directly in Properties view, use the Expression Designer, or use the Variables toolbar.
4. For visual dashboards, you can define an event action for the Label object. This means that when a user performs a certain action on the object such as a mouse click, an event such as a jump to a scene will occur.

Results

Note:
• To format line breaks in text, insert +newline+ where you want the break to occur.
• To create a border around text, display properties for the text object in the Properties view and select a line style in the LineStyle property cell.
• You can copy and paste multiple lines of text in the Properties view. The line breaks will automatically be displayed.

Inserting line objects:
You can use the line objects to draw lines, arrows, double arrows, polylines or multipolylines in your visual project.
About this task

To insert a line, arrow, double arrow, polyline or multipolyline object into your visual project:

Procedure

1. In the editor window, select the visual report page or visual dashboard scene to which you want to add the object. If the page or scene is not open in the editor, then double-click the page or scene in the Project Explorer.

2. From the Palette view, select one of the following line objects:
   - **Line**: You use this object to draw a single line. Select the Line object from the Palette view and click in the editor window (cursor has changed to a plus sign to represent drawing mode), drag the cursor to the point where you want the line to end, and release the mouse button. The line is drawn.
   - **Arrow**: You use this object to draw a single line with an arrowhead on the starting point. The Arrow object is actually a Line object with different default property values. It is drawn the same way a Line object is drawn.
   - **DoubleArrow**: You use this object to draw a single line with an arrowhead on the starting point and ending point. The DoubleArrow object is actually a Line object with different default property values. It is drawn the same way a Line object is drawn.
   - **Polyline**: You use this object to draw multiple lines that are connected at the end of each line. Select the Polyline object from the Palette view and click in the editor window (cursor has changed to a plus sign to represent drawing mode), move the cursor to the point where you want the first line to end, click the mouse button, move the cursor to the point where you want the second line to end, click the mouse button, continue until all lines are drawn. When you have finished drawing all the lines, hit the Esc (Escape) key. The cursor returns to an arrow (selection mode). All of the lines in the Polyline object are drawn.
   - **MultiPolyline**: You use this object to draw multiple polylines. Select the MultiPolyline object from the Palette view and click in the editor window (cursor has changed to a plus sign to represent drawing mode). Draw your first polyline and hit the Esc key. The cursor remains in the drawing mode. Draw your second polyline. Hit the Esc key. The cursor returns to an arrow (selection mode). Both polylines are drawn and are considered a single MultiPolyline object.

3. From the Properties view, modify the properties, such as color, width, and style to draw your line as you want. There are many properties that you can specify that will dictate the appearance of the line object or specify how the line object will behave on the visual report page or visual dashboard scene. There are several ways to specify property values for the line objects. You can select values that are listed in the Properties view, type values directly in Properties view, use the Expression Designer, or use the Variables toolbar.

4. For visual dashboards, you can define an event action for the line object. This means that when a user performs a certain action on the object such as a mouse click, an event such as a jump to a scene will occur.

Inserting shape objects:

Primitive shape objects include polygons, rectangles, rounded rectangles, and ellipses.
About this task

To insert a shape object into your visual project:

Procedure

1. In the editor window, select the visual report page or visual dashboard scene to which you want to add the object. If the page or scene is not open in the editor, then double-click the page or scene in the Project Explorer.

2. From the Palette view, select one of the following shape objects:
   - **Rectangle**: You use this object to draw a rectangle. Select the Rectangle object from the Palette view and click in the editor window (cursor has changed to a plus sign to represent drawing mode), drag the cursor to draw the rectangle, and release the mouse button. The rectangle is drawn.
   - **RoundRectangle**: You use this object to draw a rectangle that has rounded corners. Select the RoundRectangle object from the Palette view and click in the editor window (cursor has changed to a plus sign to represent drawing mode), drag the cursor to draw the rectangle, and release the mouse button. A rectangle with rounded corners is drawn.
   - **Ellipse**: You use this object to draw an ellipse. Select the Ellipse object from the Palette view and click in the editor window (cursor has changed to a plus sign to represent drawing mode), drag the cursor to draw the ellipse, and release the mouse button. The ellipse is drawn.
   - **Polygon**: You use this object to draw a closed, multi-sided figure. Select the Polygon object from the Palette view and click in the editor window (cursor has changed to a plus sign to represent drawing mode), move the cursor to the point where you want the first line of the polygon to end, click the mouse button, move the cursor to the point where you want the second line of the polygon to end, click the mouse button, continue until you have drawn the polygon shape. When you have finished drawing, select the Esc (Escape) key to terminate the drawing process. The cursor returns to an arrow (selection mode). The polygon figure is drawn.
   - **MultiPolygon**: You use this object to draw multiple polygons. Select the MultiPolygon object from the Palette view and click in the editor window (cursor has changed to a plus sign to represent drawing mode). Draw your first polygon and select the Esc key. The cursor remains in the drawing mode. Draw your second polygon. Select the Esc key. The cursor returns to an arrow (selection mode). Both polygons are drawn and are considered a single MultiPolygon object.

3. From the Properties view, modify the properties, such as color, width, and style to draw your shape object as you want. There are many properties that you can specify that will dictate the appearance of the shape or specify how the shape object will behave on the visual report page or visual dashboard scene. There are several ways to specify property values for the shape object. You can select values that are listed in the Properties view, type values directly in Properties view, use the Expression Designer, or use the Variables toolbar.

4. For visual dashboards, you can define an event action for the shape object. This means that when a user performs a certain action on the object (such as a mouse click) an event (such as a jump to a scene) will occur.

Inserting picture objects:

You use the Picture object to insert an image into your visual project.
About this task

The image that you insert into a Picture object can have one of the following formats:

- Graphics Interchange format (.gif)
- Joint Photographic Experts Group (.jpeg)
- Portable Network Graphics (.png)

To insert a picture object into your visual project:

Procedure

1. In the editor window, select the visual report page or visual dashboard scene to which you want to add the object. If the page or scene is not open in the editor, then double-click the page or scene in the Project Explorer.

2. From the Palette view, select the Picture object and click in the editor window. The picture object is drawn based on default properties. You can also select the Picture object from the Palette view, click in the editor window, drag the cursor to draw the size that you want, and release the mouse button. The picture object is drawn.

3. You must specify where the image that will be displayed in the picture object is located. You can specify that the image will be found in a query result column, in a local or network file, or embedded in the visual project. From the Properties view, click the ImageSource property cell. Click the ellipsis (…) that appears in the cell. The Picture Source window opens. Select one of the following options

   - Column to specify that the image will reside in a query result column.
     This option is available when a query is available to the visual project for selection. From the drop-down list of available queries, select the one that will return the result data. From the second drop-down list, select the column that will contain the image.
     The Queries folder in the Project Explorer view lists all the queries that are available to the visual project

   - Linked to specify that the image will reside in a local or network file. Select Linked and specify the path to the image file.

   - Embedded to specify that the image is embedded in the visual project. In order for this option to be available you must have images available to the visual project for selection. All images that are available are listed in the Images folder that is under the Globals folder in the Project Explorer. To specify that the image is embedded, select Embedded. Select the image from the drop-down list of available images.

4. From the Properties view, you can modify any of the additional properties to alter the appearance or behavior of the picture object on the visual report page or visual dashboard scene. There are several ways to specify property values. You can select values that are listed in the Properties view; type values directly in Properties view; use the Expression Designer, or use the Variables toolbar.

5. For visual dashboards, you can define an event action for the picture object. This means that when a user performs a certain action on the object such as a mouse click, an event such as a jump to a scene will occur.
Results

Inserting an alignment panel object:

The AlignmentPanel object is an invisible reference object that you can use to group and position child objects relative to a single point.

About this task

When the single reference point of the alignment panel changes, the position of all the child objects assigned to the alignment panel changes as well. An alignment panel is most useful when you want a single data point in a layout object to be represented by several objects. For example, a scatter chart usually uses a single marker object for each data point that it displays. Using an alignment panel you could specify that for each data point in a scatter chart a marker, a text label, and two buttons will be displayed in the chart. Whenever your data point value changes and is displayed in a new location, all the objects in the alignment panel are displayed in the new location as well.

To create an alignment panel and use it in your visual project:

Procedure

1. In the editor window, select the visual report page or visual dashboard scene to which you want to add the object. If the page or scene is not open in the editor, then double-click the page or scene in the Project Explorer.

2. In the Project Explorer, select the element from the visual project to which you want to add an alignment panel.

3. From the Palette view, select the AlignmentPanel object and insert it in the editor window. The AlignmentPanel object is not drawn on the canvas in the editor window. It is added as a node in the Project Explorer view.

4. On the current canvas that is in the editor window, add the graphical objects that you want to be grouped together in the alignment panel. Each object is drawn on the canvas and inserted as a node in the Project Explorer view.

5. When you have finished adding each object, select the object name in the Project Explorer, and drag the object dropping it in the AlignmentPanel node. Repeat this for each object that you want to be in the alignment panel. Upon completion, when you click any of the objects in the alignment panel, a box will surround the objects.

6. With the alignment panel selected in the Project Explorer view, select the Location cell in Properties view. Enter a value or column name. The center of the grouped objects will move to the coordinates that you specify.

Results

Note:

- In a layout object, the Location property of the AlignmentPanel object takes different values, depending on the type of layout object that you are designing. For example, in a scatter chart, the data points are x- and y-coordinates based on columns from your query. To set the location of the alignment panel in a scatter chart, you specify the column names for the x- and y-coordinates. To determine what you should set the Location property for an alignment panel to, look at the Location property of the default data symbol for that layout object.
- If the alignment panel has a single value for its Location property, you can replace it with a single column from your query. If it has more than one value
for its Location property, make sure that you replace each value with the appropriate column from your query, and that you retain the correct formatting for the Location property.

- For charts with axes that supply an alignment panel by default, the Location property for the alignment panel is set to the Pt() function which references one or more column names.

Adding control objects

Control objects are graphical objects that are used to display information or accept user input.

About this task

Each type of Control object has its own properties that make it suitable for a particular purpose. Using Control objects, you can request input from a user and based on the response to the control, you can trigger subsequent events. Control objects are only available for placement in visual dashboards.

The Controls palette offers the following control objects:

Text Box
An object that provides a box where users can enter and edit text.

Button
An object that when clicked by the user will trigger an action to be performed.

CheckBox
An object that presents two choices to the user for selection. A check box control can be either selected or clear.

RadioGroup
A composite object containing a set of radio buttons that present clickable choices to the user.

HorzSlider
A window that is used as a scrolling control for selecting an integer value within a range of values.

VertSlider
A window that is used as a scrolling control for selecting an integer value within a range of values.

Combo
An object that combines an editable text field with a list box. When displayed, the Combo object displays a box with the list of choices associated with the particular field. The user may enter additional choices using the editable text field of the Combo object.

ListBox
An object that displays to the user a list of possible values that can be selected for a particular field.

DateTime
An object that makes it easy for users to enter date or time information which can then be used by other objects.

Tree
An object that displays data objects in a visual dashboard and allows to drag these objects to the Canvas object.
PromptHierarchy
An object that allows to display the appropriate part of query results by selecting the levels and elements of a prompt hierarchy that is associated with the query.

Inserting a control object:

Control objects are graphical objects that are used to display information or accept user input.

About this task

Each type of Control object has its own properties that make it suitable for a particular purpose. Using control objects, you can request input from a user and based on the response to the control, you can trigger subsequent events. Control objects can only be inserted in visual dashboards.

To insert a Control object into a visual dashboard:

Procedure
1. In the editor window, select the visual dashboard scene to which you want to add the object. If the scene is not open in the editor, then double-click the scene in the Project Explorer. If you are inserting a control to a Level folder of a visual dashboard, then select that Level.
2. From the Palette view, select the control object from the Controls palette and insert it in the editor window.
3. From the Properties view, modify the object's properties to make the control object appear and behave as you want. There are several ways to specify property values for the control objects. You can select values that are listed in the Properties view, type values directly in Properties view, use the Expression Designer, or use the Variables toolbar.
4. You can connect the control object to a target object, in most cases that is a layout object. This allows you to change the behavior of a target layout object based on the user's input to the control object.
   For example, you can connect a ListBox control object with the department names of your company to a Grid layout object containing all the department information. Use the Connectivity tool to connect the two objects. Your user will select a department from the list box, and only the information for that department will be displayed in the grid.
5. You can define an event action for the control object. This means that when a user performs a certain action on the object such as a mouse click, an event such as a jump to a scene will occur.

Inserting a Slicer object:

A Slicer object is used to filter the multidimensional data that is displayed in a layout object.

About this task

To insert a Slicer object into a visual dashboard and set up a connection to another layout object:
Procedure

1. In the editor window, select the visual dashboard scene to which you want to add the object. If the scene is not open in the editor, then double-click the scene in the Project Explorer view.
2. From the Palette view, select the Slicer object from the Controls palette and insert it in the editor window. The Slicer Wizard opens.
3. On the first page of the wizard select the OLAP query of the layout object that you would like to filter.
4. Click Next. The Select dimension or hierarchy page opens.
5. Select the dimension of the query that you would like to filter and click Finish. The slicer object is inserted in the editor window.
6. Select the Connectivity tool from the Palette view and click the slicer object. A red dot appears.
7. Click the layout object that you want the slicer object to filter.

Results

At runtime (when the dashboard is displayed to the user), the user can click the Slicer object, and the [Dimension Name] Filter window opens. From the [Dimension Name] Filter window, the user can check which hierarchies or hierarchy elements that will be displayed in the layout object and click OK. The layout object refreshes to show only the selected data. For example, if the slicer object is set to filter the product dimension of the layout object's OLAP query, the user can choose to only display totals for a particular product.

Inserting a DateTime control object:

A DateTime control object is used to obtain date or time information from a user. The user specified date and time information can then be used to control the behavior of another object.

About this task

To insert a DateTime object into a visual dashboard and set up a connection to another object:

Procedure

1. In the editor window, select the visual dashboard scene to which you want to add the object. If the scene is not open in the editor, then double-click the scene in the Project Explorer view.
2. From the Palette view, select the DateTime object from the Controls palette and insert it in the editor window.
3. In the Property view, select the value for the Mode property of the DateTime object.
   - Select the Date mode to specify that the control object accepts the date value as input.
   - Select the Time mode to specify that the control object accepts the time value as input.
   - Select the DateTime mode to specify that the control object accepts the date and time values as input.
4. Insert the object whose behavior will be controlled by the value entered in the **DateTime** object. You can insert an object that will display the return value of the **DateTime** object or you can insert an object that will use the **DateTime** return value as a parameter.

5. To display the **DateTime** object return value:
   a. Insert a **Label** object.
   b. From the **Properties** view, select the **Text** property for the **Label** object.
   c. Open the **Expression Designer**.
   d. In the left pane of the **Expression Designer**, expand the **DateTime** object whose return value you want to display. Double-click the **Value** property.
   e. The **LabelObject.Text** property will equal the **DateTimeObject.Value**.

6. To use the **DateTime** object return value as a parameter:
   a. Create a query object (Query1) that queries a table that contains a date column. The query should include a parameter. For example: `Select * from q.interview where INTDATE=&dateparameter`. When you save this query (or move the query) to the Queries folder for the dashboard, a parameter with the same name as the substitution variable is created. In this example, a parameter called **dateparameter** is created.
   b. From the **Project Explorer**, expand the **Queries** folder, expand the Query1 folder, and right-click the parameter. Select **Edit Parameter** from the pop-up menu. When parameters are created based on their existence in a query, they are created with a datatype of literal. Change the datatype of the parameter to **DateTime**. Set a default value for the parameter. Click **OK**.
   c. Set the query parameter value to equal the return value of the **DateTime** object. From the **Project Explorer**, select the **DateTime** object whose return value will be used to set the parameter value. From the **Properties** view select the **Value** property for the **DateTime** object. Open the **Expression Designer**. In the left pane of the **Expression Designer**, expand the **Queries** folder for the dashboard. Expand the Query1 query. Expand the Parameters folder. Double-click the dateparameter parameter. The **DateTimeObject.Value** property will be set to equal the dateparameter.
   d. Insert a **Grid** object whose associated query is Query1. Query1 uses the dateparameter to control what rows are included in the grid. In this example, the dateparameter value is supplied from the **DateTime** object **Value** property.
   e. You can also use the **Connectivity** tool to set up the connection between the **DateTime** object and the **Grid** object.

**Inserting a Tree control object:**

A **Tree** object provides an additional mechanism that you can use to display objects in a visual dashboard and drag objects from a **Tree** to a **Canvas** object.

**About this task**

To insert a **Tree** object to a visual dashboard:

**Procedure**

1. Make sure the visual dashboard is opened on the **Canvas** tab of the workbench editor area.
2. Click the **Show Palette** icon to show the list of folders that hold the elements that you can add to your visual dashboard.
Note: The Show Palette icon is located in the upper-right corner of the editor area:

3. From the Controls palette of the Palette view, select the Tree object and insert it in the editor window. The Tree object is added to a visual dashboard. By default, the Tree object is named Tree1 and appears as grey box with a dotted line perimeter in the editor area:

Results

The Tree object provides a view into the hierarchical structure of data visualization objects and data transformation objects.

To view the hierarchical structure of the Tree, switch the editor to Runtime mode.

What to do next

In Runtime mode you can drag the visual dashboard objects from the Tree object to the Canvas object.

Inserting a PromptHierarchy control object:

The PromptHierarchy object is a control that allows the user to select one or more values from a flat or hierarchical list of options to be displayed.
About this task

**PromptHierarchy** control objects allow a user to control what hierarchy levels and individual elements of a prompt hierarchy are expressed in query results displays. For example, you create a table that is populated by a query that has an associated prompt hierarchy. You then create a PromptHierarchy control and link it to the query’s prompt hierarchy. Now, when you run the dashboard, you can open a window and select which parts of the query data that you want to display in the table.

To insert a PromptHierarchy control object:

**Procedure**
1. In the editor window, select the visual dashboard scene to which you want to add the object. If the scene is not opened in the editor, then double-click the page or scene in the **Project Explorer**.
2. From the **Palette** view, select the **PromptHierarchy** object and click in the editor window or double-click the **PromptHierarchy** object. The **PromptHierarchy** object is drawn based on default properties.

   **Note:** You can also select the **PromptHierarchy** object from the **Palette** view, click in the editor window, drag the cursor to draw the size that you want, and release the mouse button. The **PromptHierarchy** object is drawn.
3. Select the query that you want to connect to the control object from the **Query** drop-down list.

   **Note:** All of the queries that have been previously added to the **Queries** folder in the **Project Explorer** are available in the **Query** drop-down list.
4. Select the prompt hierarchy that you want the object to control from the **Prompt hierarchy** drop-down list.

   **Note:** All of the prompt hierarchies that have been previously linked to the queries in your **Queries** folder in the **Project Explorer** are available in the **Prompt hierarchy** drop-down list. If no prompt hierarchies have been associated with your dashboard queries, this drop-down list is empty.
5. Click **Finish**. The PromptHierarchy Wizard closes and the new PromptHierarchy control object is displayed on the canvas.

**Adding data symbol objects**

A **Data Symbol** object is an icon that is used to represent a data point in a layout object.

**About this task**

Each graphical layout object includes a default data symbol object which is used within the layout object to display the data. You can use a data symbol object to change a default data symbol in a layout object, to create a second data layer in a layout object, or independently, as a direct insert to a visual project (perhaps to create a legend for a layout).

The **Data Symbols** palette includes the following data symbol objects:

**GoogleMapMarker**

A data symbol object that displays a data point as a map marker on a Google Map object.
GoogleMapMarker is available in visual dashboards only.

Marker
A data symbol displayed as a configurable shape. The marker is typically used to represent a data point in an axis-based chart. The shape of the marker can be changed to a solid diamond, a cross, or any of several other shapes.

VerticalValueBar
A data symbol displayed as a vertical rectangle in which the height is associated with a data value returned by a query. The vertical value bar is typically used to create bar values along the y-axis of an XY chart.

VerticalLabelBar
A data symbol displayed as a rectangle that is described by a label that places it along the x-axis and a numeric value that determines its height along the y-axis.

HorizontalValueBar
A data symbol displayed as a horizontal rectangle in which the width is associated with a data value returned by a query. The horizontal value bar is typically used to create bar values along the x-axis of an XY chart.

HorizontalLabelBar
A data symbol, displayed as a rectangle that is described by a label that places it along the y-axis and a numeric value that determines its length along the x-axis.

PriceIntervalBar
A data symbol displayed as a vertical line with horizontal ticks extending to the left and right side of the line. It is used to display four numeric values over a specific time interval. The price interval bar is typically used to display the opening, closing, high, and low prices of a stock or other index on a specific date. The shape of the price interval bar can be modified to turn open/close and high/low tick marks on or off.

Candlestick
A data symbol displayed as a vertical rectangle. Used to display four numeric values over a specific time interval. The candlestick is typically used to display the opening, closing, high, and low prices of a stock on a specific date, and the color of the candlestick is designed to show gain or loss of stock value.

EventBand
A data symbol displayed as a rectangle of variable width extending from the bottom to the top of the chart. It is used to display ranges of x-values, such as a length of time an event lasted, along an axis. The event band is typically used with other data layers to depict events that correspond to other data.

ThresholdBand
A data symbol displayed as a horizontal rectangle. The threshold band is similar to the event band data symbol. It is used to display ranges of y-values along an axis and is typically used with other data layers to depict value thresholds that correspond to other series of data.

Inserting a data symbol object:
A data symbol is an icon that is used to represent a data point in a layout object.
About this task

Each graphical layout object includes a default data symbol object which is used within the layout to display the data. You can use a data symbol object to change a default data symbol in a layout object, to create a second data layer in a layout object, or independently, as a direct insert to a scene (perhaps to create a legend for a layout).

To add a data symbol to a visual project:

Procedure
1. In the editor window, select the element of your visual project to which you want to add the data symbol object. If the page, scene or object is not open in the editor, then double-click the page, scene, or object in the Project Explorer.
2. From the Palette view, select the data symbol object and insert it in the editor window.
3. From the Properties view, modify the object properties to make the data symbol appear the way you want it to.

Results

Note:
• If you want more than one object to represent each data point, first insert an alignment panel into the data template and then place the data symbol in the alignment panel.
• Adding an event band data symbol in a data layer adds depth and significance to a timeline chart. You might do this if you wanted to display particular event instances on a timeline chart that tracks a continuous trend in the market.

Adding connector objects
A connector is a graphic object, such as a straight or curved line, that provides a visual link between data points in a layout object or between graphic objects on a report page or dashboard scene.

About this task

If applicable, a layout object has a default connector which is used to provide a visual line between the data points in that layout. You can customize the line that connects all the data points in a layout by selecting a new connector from the Connectors palette. From the Connectors palette, you can also add a connection point when you want to force all data points or objects to terminate at one specified point.

The Connectors palette offers the following connectors:
• Straight Connector
  A connector between two data points or two objects that is displayed as a straight line.
• Elbow Connector
  A connector that creates a link between two data points or two objects by using both horizontal and vertical lines connected with right-angle joints.
• Spline Connector
  A connector between two data points or two objects that is displayed as a smoothly curved line.
• **Connection Point**
   
   An invisible point whose coordinates are used as a reference for a link extending from one data point to another data point.

**Inserting a connector object:**

A connector is a graphic object, such as a straight or curved line, that provides a visual link between data points in a layout object or between graphic objects on a report page or dashboard scene.

**About this task**

To insert a connector into a visual project:

**Procedure**

1. In the editor window, select the visual report page or visual dashboard scene to which you want to add the object. If the page or scene is not open in the editor, then double-click the page or scene in the Project Explorer. If you are inserting a connector into a layout, select the Layout and the Level from the Project Explorer.
2. From the Palette view, select the connector object and insert it in the editor window.
3. From the Properties view, modify the properties to draw your line.
4. Click Runtime or Preview to view the results.

**Tip:**

- To force all connection lines to connect at a single specified point, insert a ConnectionPoint and position it as you want.
- Hierarchy layouts and chart layouts with line connectors contain implicit connection points. No Connection Point object appears in the Project Explorer when these Layouts are created. However, there is a Connection Point property defined for it.
- The organization chart object uses a connection point to override the point-to-point links by making all data points connect to a single parent data point.

**Example**

Suppose you want to create a decision tree using circles to represent decisions and multiple lines to link decisions to multiple choices. You might create an organization chart and then replace the boxes in the chart with circles. To create the straight lines between the circles, you could delete the elbow connector and then insert a straight connector.

**Adding custom objects**

Custom objects are objects that you create based on one of the existing objects from the Palette view.

**About this task**

To create a custom object, you select one of the objects from the Palette view and modify its properties to meet your needs. You can then store the custom object that you created on the Custom palette where it becomes available for repeatable use.
The Custom palette is particularly useful for storing objects that are used in every scene in your dashboard or on every page of your report. For example, reusing a title object that has the font, size, frame, color, and fill that you want. This saves you time and provides consistency throughout your project.

The Custom palette is a convenient place to store your company logo as a picture object. To do so, you would create a picture object, specify your logo image as the ImageSource property for the Picture object, and then store the customized Picture object to the Custom palette.

Note: You can only store object properties with a custom object. Events and parameters associated with an object are not stored and cannot be reused from instance to instance. You must manually re-assign event actions and parameterized properties to objects if you want to duplicate those attributes.

To create a custom object and add it to the Custom palette:

Procedure
1. In the Palette view, select the object that you want to customize. For example, to create a custom text object, locate the default Label object in the palette and insert it in the editor window.
2. From the Properties view, modify the properties of the object.
3. When you are finished, return to the editor window, right-click the object and select Add to Custom Palette from the pop-up menu. The object retains the properties that you set and is placed on the Custom palette.

Results

Note:
• You can rename the objects on the Custom palette. To do so, right-click an object on the Custom palette and select Customize. The Customize Palette window opens. Enter the new name in the Name field and click OK.
• You can always change the default properties assigned to any customized object.
• The Custom palette can provide a single source for all the objects that you will use in your visual project. You can create several custom objects and place them on the Custom palette. In addition, you can place objects from any of the palettes in the Palette view on the Custom palette. That way all the objects that you will use in your project are located in one palette.

For example, if you use a Text object often in a dashboard but want the size to be 9 point rather than 12 point, and the font to be Palatino rather than Arial, you can save your customized Text object to the Custom palette. Perhaps you want to reuse a Marker object that is defined as a solid triangle shape, red color, and 9 point size; you can save the customized Marker object to the Custom palette.

Adding container objects
The Containers palette includes the EmbeddedScene, EmbeddedContent, and Canvas objects.

About this task
EmbeddedContent
An object that provides the means to add external content to a visual report or dashboard. You view the embedded content in the embedded content object.
EmbeddedScene
An object that provides the means to navigate from one scene to another. The embedded scene actually displays in the embedded scene object.

Canvas
An object that contains the visual dashboard data objects that you can manipulate in Runtime mode.

Inserting an embedded scene object:

An EmbeddedScene object is a container through which you can view all or a portion of another scene and then navigate to that scene.

About this task

You can use the EmbeddedScene object only in visual dashboards, they are not applicable to visual reports. You can embed a scene if you have already defined two scenes. The source scene is where you insert the embedded scene object. The scene specified within the embedded scene object is considered the destination scene.

To insert an embedded scene object in your visual dashboard scene:

Procedure

1. From the Project Explorer view, open the source scene in the Canvas mode.
2. From the Palette view, click the EmbeddedScene object.
3. Click in the editor window to insert the EmbeddedScene object. The Embedded scene wizard opens.
4. Select the Navigate to embedded scene when clicked with the mouse check box to automatically generate an event action for the embedded scene object. The generated event action specifies that a mouse click the embedded scene object will take a user to the embedded scene. The Navigate to embedded scene when clicked with the mouse check box should be cleared if you want to specify a different event action for the embedded scene object.
5. If you want to navigate to a location from another visual project, select the Another visual project check box and click the ellipsis (...) to the right of the field. The Open window opens.
6. Navigate to and select the visual project that you want the embedded scene to link to and click Finish. The Open window closes and control returns to the EmbeddedScene Wizard.
7. Select the name of the destination scene from the Embedded scene drop-down list.
8. Select the viewpoint of the embedded scene that you want to focus the window on in the Viewpoints list box.
9. Specify the coordinates and zoom level of the embedded scene’s canvas in from the Location fields.
10. Click OK. An EmbeddedScene object appears in the folder for your scene.

Inserting an embedded content object:

An EmbeddedContent object is a container that provides the means to add external content to a visual report or dashboard. The external content that can be displayed can be a text file or a HTML page.
About this task

To insert an **EmbeddedContent** object in your visual dashboard scene:

**Procedure**

1. From the **Project Explorer** view, open the source scene in the **Canvas** mode.
2. From the **Palette** view, click the **EmbeddedContent** object.
3. Click in the editor window to insert the **EmbeddedContent** object. The **EmbeddedContent** object is drawn based on default size properties. Click in the editor window and drag the cursor to insert the **EmbeddedContent** object drawn to your specific size. The Embedded content wizard opens.
4. In the **EmbeddedContent mode** area click **ExternalData** to add a separate file or **DirectData** to add the content as a string.
5. If you selected **ExternalData** specify the location of the file you want to add in the **Content location** field. You can click the ellipsis (…) to search for the location.
6. Specify the type of content that will be embedded in the **Content type** field. Select **Text** to display content in text format and **HTML** to display content in HTML format.
7. If you selected **Text** content type, select the required character encoding from the **Character encoding** list.
8. If you selected **DirectData**, click the ellipsis (…) to add the required content in the **Content** field, and click **OK**.
9. Specify the type of content that will be embedded in the **Content type** field. Select **Text** to display content in text format and **HTML** to display content in HTML format.
10. If you selected **Text** content type, select the required character encoding from the **Character encoding** list.
11. Click **OK**. An **EmbeddedContent** object is created. An entry is made for the object in the **Project Explorer** view.

**Results**

**Note:** Embedded content objects support only a limited amount of HTML tags in the Flash rendering runtime mode.

**Inserting a Canvas object:**

A **Canvas** object contains the visual dashboard data objects that you can manipulate in **Runtime** mode.

**About this task**

To insert a **Canvas** object in a visual dashboard:

**Procedure**

1. Make sure the visual dashboard is opened on the **Canvas** tab of the workbench editor area.
2. Click the **Show Palette** icon to show the list of folders that hold the elements that you can add to your visual dashboard.

**Note:** The **Show Palette** icon is located in the upper-right corner of the editor area:
3. From the **Containers** palette of the Palette view, select the **Canvas** object and insert it in the editor window. The **Canvas** object is added to a visual dashboard. By default, the **Canvas** object is named **Canvas1** and appears as *grey box with a dotted line perimeter* in the editor area:

![Canvas1](image)

**Results**

The Canvas object has an internal tree. The repository subtree or queries subtree of data visualization objects and data transformation objects located in visual dashboard depends on the **Source** property of the internal tree.

The Canvas object has an internal tree that represents (in its entirety) the hierarchical structure of data visualization objects and data transformation objects. Keep in mind that Canvas supports only a subset of data visualization objects and data transformation objects.

To view the internal tree of the Canvas, switch the editor to **Runtime** mode.

**Canvas types and layouts:**

Each Canvas type maintains a specific layout that depends on the objects that the Canvas contains. The layout of a Canvas is determined by the data objects placed on it and the size and location of the Canvas in the visual dashboard.

There are three types of Canvases:

**Built-in**

Built-in Canvas is stored inside the visual dashboard and is created while designing the dashboard with Canvas.

Built-in Canvas contains the objects and layouts that are the part of a visual dashboard. Typically, a Built-in Canvas represents what the designer believes to be the most user-appropriate set of data objects, as well as the layout of those objects.

The designer of the visual dashboard can also set the default type to a Built-in Canvas at design time. This default type is loaded for the users. If the designer of the visual dashboard does not set the default type, the users see the last canvas state that was used by the designer.
Users can adopt a Built-in Canvas and use it as a building block for their visual dashboards, tailoring it to suit the needs of their visual dashboard and then saving it under a new name. The tailored Canvas would be accessible, alongside those that come as part of the visual dashboard.

**Shared**

Shared Canvas is stored in the unified shared space in repository.

Shared Canvas is stored in a shared repository when the visual dashboard on which the canvas resides is open in *Runtime* mode. All repository users can access and use a Shared Canvas.

**Personal**

Personal Canvas is stored in the user home workspace.

Personal Canvas is accessed by users while the visual dashboard is in *Runtime* mode and stored in an existing personal workspace. If a personal workspace does not exist, users cannot save the canvas as a Personal Canvas. Each user can also set default type for current canvas that will be loaded for this user in the future.

Each object on the Canvas includes the following attributes:

- Key
- Container location and size
- Object layout (each object has to be able to save it’s current layout).

*Adding objects to the Canvas:*

You can add objects to a Canvas to provide functionality to your visual dashboard.

**About this task**

Before adding objects to the Canvas, make sure:

- You created a visual dashboard and added a Canvas object to the dashboard.
- You placed the Canvas to the required location on the visual dashboard and sized the Canvas object appropriately.
- You set the Canvas properties.

The editor must be in *Runtime* mode.

*Adding objects to the Canvas in the Flash mode:*

You can add objects to the Canvas object of a visual dashboard in the Flash mode.

**About this task**

To add objects to the Canvas in the Flash mode:

**Procedure**

1. Click the *Runtime* tab in the editor to place the visual dashboard in *Runtime* mode.
2. Drag your cursor on to the Canvas object.
   
   The Canvas changes color from white to gray and there is a cross in the center of the Canvas indicating the position of the first display.
Single clicking anywhere in the grey display area brings up the Canvas's Internal Tree.

Single clicking a query and clicking OK or double-clicking a query, runs that query and displays the result in the default format for the query.

3. Drag objects from Tree component to the Canvas.
   If an object is not supported by Canvas, the system does not allow you to drag & drop the object.

4. Left-click the Cross icon to show the list of objects.

5. Click the Refresh Tree button to refresh the content of workspaces in the tree.

6. Locate the object that you want to add to the Canvas, select it and click OK.
   Currently, Canvas supports the following objects:
   - Query
   - Visual Query
   - Analytical Query
   - Dynamart
   - Compound Dynamart
   - Table
   - Drill-down Path

Note: You can add visual queries with prompts to the Canvas. Use local or global parameters to manage prompts automatically. Insert a parameter to the visual project, make sure that it has the same name as one of the prompted variables used in the visual query, and then set the default value. This value is set for the prompt variable when you add the visual query to the Canvas.

Results

You have added objects to the Canvas.
What to do next

After adding an object to the Canvas, the default Display Mode for that object is rendered graphically. For example:

Display Mode for chart object

Display Mode for query object

Click any of the widgets listed on an object’s Display Mode and the image is rendered accordingly on the Canvas.

The buttons on the toolbar vary depending on the type of query. The complete set of widgets allow for the following actions:

• Switching display mode from chart to grid
• Changing chart type to 2D Charts:
  – Area
  – Bar
  – Bubble
  – Column
  – Grid
  – Line
  – Pie
  – Scatter
  – Speedometer
• Changing chart type to 2D Charts with Depth:
  – Area (except stacked Area charts)
  – Bar
  – Column
  – Pie
• Changing the query columns and aggregation being charted.
  Numeric columns can have an aggregation value of:
  – None
  – Count
  – Maximum
  – Minimum
  – Sum
  – Average
  Character columns can have an aggregation value of:
  – None
  – Count
- Viewing the information about the source object that is added to the Canvas
- Deleting the query result from the Canvas

Adding objects to the Canvas in the HTML5 mode:

You can add objects to your Canvas of your visual dashboard in the HTML5 mode.

About this task

To add objects to the Canvas in the HTML5 mode:

Procedure
1. Click the Runtime tab in the editor area to place the visual dashboard in Runtime mode.

2. Click to expand the menu of the Canvas object.

The Canvas with its tree opens.

3. In the right tree pane, navigate to the object you want to add and the drag it to the Canvas. The object is run and the result is displayed in the default format.

Currently, Canvas supports the following objects:
- Analytical Query
- Compound Dynamart
- Drill-down Paths
- Dynamart
- Query
- Table
- Visual Query

Note: You can add queries with prompts to the Canvas. Use local or global parameters to manage prompts automatically. Insert a parameter to the visual project, make sure that it has the same name as one of the prompted variables used in the query, and then set the default value. This value is set for the prompt variable when you add the query to the Canvas.
Results
You have added objects to the Canvas.

What to do next
After adding an object to the Canvas, the default Display Mode for that object is rendered graphically. For example:

Display Mode for charts

Display Mode for query object

Click any of the widgets listed on an object's Display Mode and the image is rendered accordingly on the Canvas.

The buttons on the toolbar vary depending on the type of query. The complete set of widgets allow for the following actions:

• Switching display mode from chart to grid ( )

• Changing chart type to 2D Charts ( ):
  – Area
  – Bar
  – Bubble
  – Column
  – Grid
• Changing chart type to **2D Charts with Depth**:
  - Area (except stacked Area charts)
  - Bar
  - Column
  - Pie

• Changing the query columns and aggregation being charted:
  Numeric columns can have an aggregation value of:
  - None
  - Count
  - Maximum
  - Minimum
  - Sum
  - Average
  - First value
  - Last value
  - Standard deviation
  Character columns can have an aggregation value of:
  - Count

• Editing the title of the added object

• Viewing the information about the source object that is added to the Canvas

• Deleting the query result from the canvas

*Managing Canvas operations using the Canvas toolbar:*

The Canvas toolbar provides actions for managing the Canvas in ad hoc reports and visual dashboards.
Canvas toolbar options

A toolbar displays whenever you add an object to the Canvas:

- In the Flash mode:

- In the HTML5 mode:

Each option on the toolbar is described in the following table:

<table>
<thead>
<tr>
<th>Toolbar widget</th>
<th>Flash mode</th>
<th>HTML5 mode</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Select" /></td>
<td><img src="image" alt="Select" /></td>
<td></td>
<td>Select Create Layout button to create an empty layout.</td>
</tr>
<tr>
<td><img src="image" alt="New Layout" /></td>
<td><img src="image" alt="New Layout" /></td>
<td></td>
<td>Select New Layout options from the list for managing existing Canvas layout.</td>
</tr>
<tr>
<td><img src="image" alt="Save Layout" /></td>
<td><img src="image" alt="Save Layout" /></td>
<td></td>
<td>Click the Save Layout as button to save the Canvas as a Built-in, Shared or Personal Canvas type.</td>
</tr>
<tr>
<td><img src="image" alt="Save Layout" /></td>
<td><img src="image" alt="Save Layout" /></td>
<td></td>
<td>Click the Save Layout button to save the Canvas.</td>
</tr>
<tr>
<td><img src="image" alt="Select" /></td>
<td><img src="image" alt="Select" /></td>
<td></td>
<td>Select New Layout &gt; Open default layout.</td>
</tr>
<tr>
<td><img src="image" alt="Select" /></td>
<td><img src="image" alt="Select" /></td>
<td></td>
<td>Click the Open Default Layout button to switch to default Canvas layout.</td>
</tr>
<tr>
<td><img src="image" alt="Select" /></td>
<td><img src="image" alt="Select" /></td>
<td></td>
<td>Click the Set Layout as Default button to make the current layout.</td>
</tr>
<tr>
<td><img src="image" alt="Select" /></td>
<td><img src="image" alt="Select" /></td>
<td></td>
<td>Click the Clear Layout button to remove the object.</td>
</tr>
<tr>
<td><img src="image" alt="Select" /></td>
<td><img src="image" alt="Select" /></td>
<td></td>
<td>Click the Switch to Fullscreen Mode button to maximize the Canvas.</td>
</tr>
<tr>
<td><img src="image" alt="Select" /></td>
<td><img src="image" alt="Select" /></td>
<td></td>
<td>Click the Switch to Normal Mode button to minimize the Canvas.</td>
</tr>
</tbody>
</table>
### Table 28. Toolbar options (continued)

<table>
<thead>
<tr>
<th>Toolbar widget</th>
<th>Flash mode</th>
<th>HTML5 mode</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Flash mode" /></td>
<td><img src="image" alt="HTML5 mode" /></td>
<td>There is no such option in the HTML5 mode.</td>
<td>Click <strong>Pin/Unpin Toolbar</strong> button to pin or unpin the Canvas toolbar.</td>
</tr>
<tr>
<td><img src="image" alt="Pin/Unpin Toolbar" /></td>
<td><img src="image" alt="Wiring" /></td>
<td>Click the <strong>Wiring</strong> or <strong>Toggle Wiring Lines</strong> button to activate the wiring mode.</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Wiring or Toggle Wiring Lines" /></td>
<td><img src="image" alt="Export Layout" /></td>
<td>Click the <strong>Export Layout</strong> button to open the export pane and select the available export options:</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td><img src="image" alt="Export to PDF" /></td>
<td>Export the current content of the Canvas to PDF.</td>
</tr>
<tr>
<td></td>
<td></td>
<td><img src="image" alt="Export to PNG" /></td>
<td>Export the current content of the Canvas to PNG.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Print the current content of the Canvas.</td>
<td></td>
</tr>
<tr>
<td>The separate <strong>Refresh Tree</strong> button is available in Flash.</td>
<td><img src="image" alt="Refresh Tree" /></td>
<td>Click the <strong>Refresh Tree</strong> button to refresh the content of workspaces in the tree. In Flash, the button appears when you expand the canvas internal tree for adding an object.</td>
<td></td>
</tr>
</tbody>
</table>

**Wiring objects on the Canvas:**

Wiring objects defines the relationship and dependencies between objects. Use the object Wiring Mode to connect objects on the Canvas.

**About this task**

The following topics describe wiring in the Flash and HTML5 modes.

**Wiring objects on the Canvas in Flash:**

Use the object Wiring Mode to connect objects on the Canvas.
About this task

To wire objects in Flash, perform the following procedure:

Procedure

1. Click the **Wiring** icon on the Canvas Toolbar to switch the Canvas to the **Wiring Mode**.

2. On the Canvas, select the source object and the target object.
   
   The source object is highlighted with blue and the target object is highlighted with green.
   
   When you wire objects, the system checks for possibility of creating a new link. You can select any data object that supports wiring on your Canvas as the source object. However, once the source object has been set, only those objects that are valid targets to the source are available (highlighted) for wiring.
   
   **Note:** Any data object that supports wiring can be the source object and the target object at the same time. The number of links is limited only by the number of available columns. Each column can be the source in several links and the target in the only one.
   
   **Note:** You can also wire objects on the Canvas by using drag-and-drop method. When you drag an object the **Wiring** icon is displayed over the canvas.
   
   The Create a Link window opens.

3. In the Create a Link window, specify the source and destination columns.
   
   From the **Source** list, select a column of the source object.
   
   From the **Target** list, select the column of the target object
   
   If the queries already contain specified output and input columns, these columns are displayed and automatically selected. Otherwise, all the columns from the result set are displayed.

4. Click **Create** to wire the objects to one another and to create the link.

Results

You have wired objects on the Canvas.

In objects that are linked, the target object displays the data according to the selected element of the source object.

When you select another element, the data that displays changes. The selected element is highlighted. The second click the same element deselects it. In this case, the target object displays the result as if all the elements of the source object were selected.

**Wiring objects on the Canvas in HTML5:**

Use the object Wiring Mode to connect objects on the Canvas in HTML5.
About this task

Wiring objects defines the relationship and dependencies between objects.

To wire objects in HTML5, do the following actions:

Procedure

1. Open the canvas toolbar and select the source object and the target object that you want to wire.
   The system checks for possibility of creating a new link. The Create Wiring buttons of objects that can be wired are highlighted in green.

   Note: Any data object that supports wiring can be the source object and the target object at the same time. The number of links is limited only by the number of available columns. Each column can be the source in several links and the target in the only one.

2. Click ( ) of the first object and drag it to ( ) of the second object. The Create Link window opens.

3. In the Create Link window, specify the source and destination columns.
   From the Source column list, select a column of the source object.
   From the Target column list, select the column of the target object.
   If the queries already contain specified output and input columns, these columns are displayed and automatically selected. Otherwise, all the columns from the result set are displayed.

4. Click Create to wire the objects to one another and to create the link.

5. Click the Toggle Wiring Lines icon ( ) on the Canvas Toolbar to turn on or turn off the wiring of these objects on the Canvas.

   Tip: When wiring is turned on, the wired objects are joined with a green wiring line.

Editing linked objects:

Use the Edit Link function to make changes to the source and target objects or to remove the link relationship between the objects.

About this task

To edit linked objects, perform the following procedure:

Procedure

1. From the Canvas, select the source object and the target object by clicking them. The Edit the Link window opens.

2. In the Edit the Link window you can edit the source columns of the source object and destination columns of the target object in the corresponding fields.

3. To remove the linkage between the two objects, click Delete. The link between the objects is removed and the Edit the Link window closes.
4. Click OK to save the changes and close the Edit the Link window.
5. Click Cancel to discard the changes and close the Edit the Link window.

Managing input and output columns for the visual query:

You can specify available input and output columns for the current visual query. These columns will be used for wiring objects on a canvas when creating visual dashboards or ad hoc reports.

About this task

The steps in this procedure describe how to specify input and output columns for a visual query.

Procedure

1. Select Query > Wiring Settings to open the Wiring Settings window.
2. In the Wiring Settings window the list of available columns is displayed. To specify input and output columns select the corresponding check boxes.
3. After you have specified the input and output columns, click OK to apply the changes and close the Wiring Settings window.

Working with layout objects

The Layouts palette contains graphical objects that you can use to display the data that has been obtained as a result of a query. Layout objects include a wide assortment of charts, graphs, maps, tables and grids.

About this task

The following chart layouts are available:

- BarChart
- PieChart
- ColumnChart
- ScatterChart
- XYChart
- StockChart
- CandlestickChart
- TimelineChart
- EventBandChart
- MultivariateChart
- LinearMap

The following hierarchy layouts are available for showing dependency relationships:

- Cluster graph
- Organization chart
- Tree chart

The following pattern layouts are available for showing data in an ordered pattern but without quantity or dependency relationships:

- Table
- Grid
• SimpleForm
• Matrix
• Spiral
• Horizon

Also, the Google Map object can be used in Flash-based dashboards to display result sets as geographic data points.

The SolidChart object is a monolithic object. It can be used to display your query results data in a visually rich format, like charts for queries.

Inserting a table:

Table objects let you display query results in a simple table.

About this task

To insert a Table object:

Procedure
1. Double-click the Table object in the Layouts palette of the Palette view. The Table Wizard opens.
2. Specify whether you want to populate the table with data from an existing query and click Next.
3. If you specified No, specify the table options and click Finish. The wizard closes and the table is displayed in the editor window.
4. If you specified Yes, select the query that you want to use to populate the table and click Next.
5. Select the columns that you want to include in the table and click Next.
6. Specify the table options and click Next.
7. Specify paging options and click Finish. The wizard closes and the table is displayed in the editor window.

Inserting a simple table:

You can insert a simple table object that is not populated with data from an existing query.

About this task

To insert a simple Table object:

Procedure
1. Double-click the Table object in the Layouts palette of the Palette view. The Table Wizard opens.
2. Select No and click Next. The Specify table options page opens.
3. Specify the number of rows the table will have in the Rows list.
4. Specify the number of columns the table will have in the Columns list.
5. Specify the alignment of the text in the cells in the Alignment drop-down list. The valid options are:
   • Left
   • Right
• Center

6. Specify the minimum width of each cell in the Minimum width field.
7. Click Finish. The wizard closes and the table is displayed in the editor window.

Inserting a populated table:

You can insert a table object that is populated with data from an existing query.

About this task

To insert a populated Table object:

Procedure

1. Double-click the Table object in the Layouts palette of the Palette view. The Table Wizard opens.
2. Select Yes and click Next. The Select the query page opens.
3. In the All Data Sources tree, navigate to the table or query that will be run to obtain the layout object’s query result set and click Next.
4. Select one or more columns in the Available fields list box and click Move Right to move them to the Columns to be created list box. This field lists the names of the columns of data that are included in the selected query result set. The Specify table options page opens.
5. Select the Include title row check box to create a title row.
6. Select the Include header row check box to create a header row.
7. Select the Include footer row check box to create a footer row.
8. Select the Word-wrap cell text check box to make the text in each cell fit the cell space.
9. Specify the alignment of the text in the cells in the Alignment list. The valid options are:
   • Left
   • Right
   • Center
10. Specify the minimum width of each cell in the Minimum width field.
11. Specify the height limit of the table in the Height Limit field. Zero (0) value means that the table height is unlimited. Any value different from zero (0) sets the maximum table height in Flash and HTML5 rendering modes.
12. Select the necessary Scroll Scope value from the list:
   • Entire Table to scroll the whole table.
   • Data Only to scroll only table data.
   • Data and Heading to scroll only table data and column headers.
13. Select the necessary value from Vertical Scroll list to set the vertical scrolling.
14. Specify the width limit of the table in the Width Limit field. Zero (0) value means that the table width is unlimited. Any value different from zero (0) sets the maximum table width only in Flash and HTML5 rendering modes.
15. Click Next. The Specify paging options page opens.
16. Select the Enable paging check box to enable paging. Paging allows you to display only a certain amount of rows at a time.
17. Specify the amount of rows that you want to display at a time in the Rows per page list.
18. Select the first row in the result set to display in the **First displayed row** list.

19. Select the **Add navigation bar** check box to automatically create page forward and page backwards buttons that control paging actions.

20. Click **Finish**. The wizard closes and the table is displayed in the editor window.

Including a summary row:

You can include a summary row in a table using the **Show Total Summaries** menu item.

About this task

By selecting the **Show Total Summaries** command in the associated query's **Results** menu, you can include a summary row with aggregated totals in the table object. To include a summary row in a table:

**Procedure**

1. Insert a table object that is populated with data from an existing query into the dashboard canvas.

2. Open the **Queries** folder in the **Project Explorer** and double-click the table object's associated query. The query opens in the query editor.

3. Run the query. The query results are displayed in the query results editor.

4. Select **Results > Show Total Summaries**. A summary row is added to the query results grid.

5. Right-click a column header label, select **Grouping and Aggregation**, and select the aggregated total that you want to add to the summary row. Repeat this for any column that you want to add an aggregated total to.

6. Save and close the query.

7. Run the visual dashboard. The summary row is displayed, with the aggregated totals that you selected, in the table object's grid.

Formatting a summary row:

By using the **isSummaryRow()** function, you can add formatting to a table object's summary row.

About this task

The **isSummaryRow()** expression designer function checks whether a row in a table object's grid is a normal row of the result set, or the summary row of the result set. If the row is the summary row, it returns true. Using this functionality, you can add specific formatting to a table object's summary row. To add formatting to a table object's summary row:

**Procedure**

1. In the **Project Explorer**, navigate to the table object's data template.

2. To exclude the summary row, select the table row in the data template and click the **Value** field of the **Visible** property in the **Properties** view.

3. Click the **Edit with Expression Designer** button on the **Properties** view toolbar. The Expression Designer opens.
4. Delete the text in the field and double-click the if function in the Logical function list. =if(bConditional, vIfTrue, vIfFalse) is inserted into the field with bConditional highlighted.

5. Double-click the isSummaryRow function in the Information function list. isSummaryRow() is inserted into the function.

6. Replace vIfTrue with false and vIfFalse with true.

7. Click OK. The Expression Designer closes.

8. To add formatting to the summary row's cells or content, select the table cell or table cell label in the data template and click the Value field of the property that you want to format in the Properties view.

9. Click the Edit with Expression Designer button on the Properties view toolbar. The Expression Designer opens.

10. Delete the text in the field and double-click the if function in the Logical function list. =if(bConditional, vIfTrue, vIfFalse) is inserted into the field with bConditional highlighted.

11. Double-click the isSummaryRow function in the Information function list. isSummaryRow() is inserted into the function.

12. Replace vIfTrue with the formatting that you want to apply to the summary row of the query result set. Replace vIfFalse with the formatting that you want to apply to the body rows of the query result set.

13. Click OK. The Expression Designer closes.

Inserting a grid:

The Grid object displays query results in a simple grid.

About this task

To insert a grid:

Procedure
1. Double-click the Grid object in the Layouts palette of the Palette view.
2. In the Grid Wizard, navigate to the table or query that will provide data for the Grid object.
3. Click Finish. The Grid Wizard closes and the Grid object is displayed on the scene.

Inserting a bar or column chart:

Bar charts and column charts display data as a series of bars on an x or y axis.

About this task

To insert a bar or column chart:

Procedure
1. Double-click the BarChart or ColumnChart object in the Layouts palette of the Palette view. The BarChart Wizard or ColumnChart Wizard opens.
2. In the All Data Sources tree, navigate to the table or query that will be run to obtain the layout object's query result set and click Next. The Select layout type page opens.
3. Click **Standard** and click **Next**. The Select the field(s) to be displayed as bars page opens.

4. Select one or more columns in the **Available fields** list box and click **Move Right** to move them to the **Fields to be displayed** list box. This field lists the names of the columns of data that are included in the selected query result set. The columns that you move to the **Fields to be displayed** list box will be displayed as the bars along a labeled axis in the column or bar chart.

5. Click **Next**. The Set up labels and legend information for chart page opens.

6. Select the **Include a label** check box to include a label that displays for each value bar in the chart.

7. Specify a data column in the **Label field** drop-down list that will be used to populate the labels for each value bar in the chart. This field is only valid if you selected the **Include a label** check box. For example, if you select a column listing salaries in the **Fields to be displayed** list box, and a column listing the names of employees in the **Label field** drop-down list, the chart will display the name of each person next to value bars that show their individual salaries.

8. Select the **Display the value of each bar** check box to display the data value at the end of each value bar.

9. Select the **Stack bars when there are multiple series** check box to stack value bars of different data series on the same line. The value bars representing the first data column in the **Fields to be displayed** list box are drawn first, and subsequent value bars start where the ones before them end. For example, if bar 1 has a value of 50 and bar 2 has a value of 20, bar 1 is displayed from 0 to 50 and bar 2 is displayed from 50 to 70.

10. Click **Next**. The Specify paging options page opens.

11. Select the **Enable paging** check box to enable paging. Paging allows you to display only a certain amount of rows at a time. For instance, if you only display ten rows at a time, only ten columns or bars will be displayed for each page.

12. Select the amount of rows that you want to display at a time in the **Rows per page** list.

13. Select the first row in the result set to display in the **First displayed row** list.

14. Select the **Add navigation bar** check box to automatically create page forward and page backwards buttons that control paging actions.

15. Click **Finish**. The wizard closes, and the bar or column chart is displayed in the editor window.

**Inserting an extended bar or column chart:**

Extended bar and column charts allow you to roll up data across one or more columns in a result set.

**About this task**

To insert an extended bar or column chart:

**Procedure**

1. Double-click the **BarChart** or **ColumnChart** object in the **Layouts** palette of the **Palette** view. The BarChart Wizard or ColumnChart Wizard opens.

2. In the **All Data Sources** tree, navigate to the table or query that will be run to obtain the layout object’s query result set and click **Next**. The Select layout type page opens.
3. Click **Extended** and click **Next**. The Select the field(s) to be displayed as bars page opens.

4. Select one or more columns in the **Available fields** list box and click **Move Right** to move them to the **Field** column of the **Fields to be displayed** table. This field lists the names of the columns of data that are included in the selected query result set. The columns that you move to the **Fields to be displayed** table will be displayed as the bars along a labeled axis in the column or bar chart.

5. Click a cell in the **Aggregation** column of the **Fields to be displayed** table. A drop-down list opens. Choose an option to specify how the column is aggregated.

6. Click **Next**. The Select one or more fields to be moved in the side group page opens.

7. Select one or more columns in the **Available fields** list box and click **Move Right** to move them to the **Fields to be moved** list box. The columns that you move to the **Fields to be moved** list box will be rolled up in the query result set, and the order of the columns will determine the roll up hierarchy.

8. Click **Next**. The Set up labels and legend information for chart page opens.

9. Select the **Display the value of each bar** check box to display the data value at the end of each value bar.

10. Select the **Stack bars when there are multiple series** check box to stack value bars of different data series on the same line. The value bars representing the first data column in the **Fields to be displayed** list box are drawn first, and subsequent value bars start where the ones before them end. For example, if bar 1 has a value of 50 and bar 2 has a value of 20, bar 1 is displayed from 0 to 50 and bar 2 is displayed from 50 to 70.

11. Click **Finish**. The wizard closes, and the bar or column chart is displayed in the editor window.

**Displaying OLAP data in a bar or column chart:**

You can display OLAP data in a bar or column chart by using a multidimensional query to obtain query results.

**About this task**

Bar or column charts that are run using multidimensional queries allow you to roll up data across one or more columns in the associated query result set. This allows users to drill down into specific data symbols to obtain only the data they need, without having to view the entire result set at once.

To insert a bar or column chart that displays OLAP data:

**Procedure**

1. Double-click the **BarChart** or **ColumnChart** object in the **Layouts** palette of the **Palette** view. The BarChart Wizard or ColumnChart Wizard opens.

2. In the **All Data Sources** tree, navigate to an OLAP query in the **Multidimensional Data Sources** folder that will be run to obtain the layout object’s query result set and click **Next**. The Select the field(s) to be displayed as bars page opens.

3. Select one or more columns in the **Available fields** list box and click the **Move Right** button to move them to the **Fields to be displayed** list box. This field lists the names of the columns of data that are included in the selected query.
result set. The columns that you move to the **Fields to be displayed** list box will be displayed as the bars along a labeled axis in the column or bar chart.

4. Click **Next**. The Select dimension or hierarchy page opens.

5. Select one or more dimensions of the cube model that you want to use to express the query results and click **Next**. Select more than one dimension by holding the **Ctrl** key and clicking the dimensions that you would like to use. The Set up labels and legend information for chart page opens.

6. Select the **Display the value of each bar** check box to display the data value at the end of each value bar.

7. Select the **Stack bars when there are multiple series** check box to stack value bars of different data series on the same line. The value bars representing the first data column in the **Fields to be displayed** list box are drawn first, and subsequent value bars start where the ones before them end. For example, if bar 1 has a value of 50 and bar 2 has a value of 20, bar 1 is displayed from 0 to 50 and bar 2 is displayed from 50 to 70.

8. Click **Finish**. The wizard closes, and the bar or column chart is displayed in the editor window.

**Inserting a pie chart using a relational query:**

Pie charts display data in a circular chart that is cut into wedges.

**About this task**

To insert a pie chart using a relational query:

**Procedure**

1. Double-click the PieChart object in the Layouts palette of the Palette view. The PieChart Wizard opens.

2. In the All Data Sources tree, navigate to the table or relational query that will be run to obtain the layout object's query result set and click **Next**. The Select layout type page opens.

3. Click **Standard** and click **Next**. The Select options for the pie chart page opens.

4. In the **Wedge values** drop-down list, specify the name of the column of data in your query result set that will contain the values for each pie wedge in the pie chart. You must select a data column from the drop-down list of available data columns in your query result set.

5. Select the **Include a label for the wedge** check box to include a label for each wedge.

6. In the **Label field** drop-down list, specify the data column in your query result set that will contain the label information for each pie wedge. You must select a field from the drop-down list of available data columns from your query result set if you selected the **Include a label for the wedge** check box.

7. Select the **Include the value of the wedge** check box to display in the pie chart the value of each wedge.

8. Select the **Include the percentage the value represents of the whole pie** check box to display a percentage value for each pie wedge.

9. Click **Finish**. The PieChart Wizard closes and the chart is displayed in the editor window.
Inserting an extended pie chart:

Extended pie charts allow you to roll up data across one or more columns in a result set.

About this task

To insert an extended pie chart:

Procedure
1. Double-click the PieChart object in the Layouts palette of the Palette view. The PieChart Wizard opens.
2. In the All Data Sources tree, navigate to the table or query that will be run to obtain the layout object's query result set and click Next. The Select layout type page opens.
3. Click Extended and click Next. The Select one or more fields to be moved in the side group page opens.
4. Select one or more columns in the Available fields list box and click Move Right to move them to the Fields to be moved list box. The columns that you move to the Fields to be moved list box will be rolled up in the query result set, and the order of the columns will determine the roll up hierarchy.
5. Click Next. The Select options for the pie chart page opens.
6. In the Wedge values drop-down list, specify the name of the column of data in your query result set that will contain the values for each pie wedge in the pie chart. You must select a data column from the drop-down list of available data columns in your query result set.
7. In the Aggregation drop-down list, specify how the column of data specified in the Wedge values field will be aggregated.
8. Select the Include a label for the wedge check box to include a label for each wedge.
9. In the Label field drop-down list, specify the data column in your query result set that will contain the label information for each pie wedge. You must select a field from the drop-down list of available data columns from your query result set if you selected the Include a label for the wedge check box.
10. Select the Include the value of the wedge check box to display in the pie chart the value of each wedge.
11. Select the Include the percentage the value represents of the whole pie check box to display a percentage value for each pie wedge.
12. Click Finish. The PieChart Wizard closes and the chart is displayed in the editor window.

Displaying OLAP data in a pie chart:

You can display OLAP data in a pie chart by using a multidimensional query to obtain query results.

About this task

Pie charts that are run using multidimensional queries allow you to roll up data across one or more columns in the associated query result set. This allows users to drill down into specific data symbols to obtain only the data they need, without having to view the entire result set at once.
To insert a pie chart that displays OLAP data:

**Procedure**

1. Double-click the **PieChart** object in the **Layouts** palette of the **Palette** view. The PieChart Wizard opens.
2. In the **All Data Sources** tree, navigate to an OLAP query in the **Multidimensional Data Sources** folder that will be run to obtain the layout object’s query result set and click **Next**. The Select dimension or hierarchy page opens.
3. Select one or more dimensions of the cube model that you want to use to express the query results and click **Next**. Select more than one dimension by holding the Ctrl key and clicking the dimensions that you would like to use. The Select options for the pie chart page opens.
4. In the **Wedge values** drop-down list, specify the name of the column of data in your query result set that will contain the values for each pie wedge in the pie chart. You must select a data column from the drop-down list of available data columns in your query result set.
5. Select the **Include a label for the wedge** check box to include a label for each wedge.
6. In the **Label field** drop-down list, specify the data column in your query result set that will contain the label information for each pie wedge. You must select a field from the drop-down list of available data columns from your query result set if you selected the **Include a label for the wedge** check box.
7. Select the **Include the value of the wedge** check box to display in the pie chart the value of each wedge.
8. Select the **Include the percentage the value represents of the whole pie** check box to display a percentage value for each pie wedge.
9. Click **Finish**. The PieChart Wizard closes and the chart is displayed in the editor window.

**Inserting a multivariate chart:**

Multivariate charts display data in a grid of scatter charts.

**About this task**

To insert a multivariate chart:

**Procedure**

1. Double-click the **MultiVariateChart** object in the **Layouts** palette of the **Palette** view. The MultiVariateChart Wizard opens.
2. In the **All Data Sources** tree, navigate to the table or query that will be run to obtain the layout object’s query result set and click **Next**. The Select the fields to be mapped to axes page opens.
3. Select one or more columns in the **Available fields** list box and click the **Move Right** button to move them to the **Axes to be created** list box. This field lists the columns that have been selected from your query result set whose data will be mapped to an axis in the multivariate chart. The axes of the multivariate chart are labeled along the diagonal, with the first axis drawn in the lower left grid cell using the first column of data specified in the **Axes to be created** list box. Each column of data is charted along the diagonal in the order that it appears in the **Axes to be created** list with the last axis drawn in the upper right grid cell.
4. Click **Next**. The Specify multivariate chart options page opens.
5. Specify the width of each cell in the **Width** field.
6. Specify the Height of each cell in the **Height** field.
7. Specify the gutter width (space between cells horizontally) in the **Horizontal gutter** field.
8. Specify the gutter height (space between cells vertically) in the **Vertical gutter** field.
9. Click **Finish**. The MultiVariateChart Wizard closes and the chart is displayed in the editor window.

**Inserting a scatter or XY chart:**

Scatter charts display data in a two-dimensional graph in rectangular coordinates. XY charts display data in a two-dimensional graph consisting of connected points whose coordinates represent two variables.

**About this task**

To insert a scatter or XY chart:

**Procedure**

1. Double-click the **ScatterChart** or **XYChart** object in the **Layouts** palette of the **Palette** view. The ScatterChart Wizard or XYChart Wizard opens.
2. In the **All Data Sources** tree, navigate to the table or query that will be run to obtain the layout object's query result set and click **Next**. The Select the field(s) to be displayed as **Y** values page opens.
3. Select one or more columns in the **Available fields** list box and click the **Move Right** button to move them to the **Fields to be displayed** list box. This field lists the names of the columns of data that are included in the selected query result set. The columns that you move to the **Fields to be displayed** list box will be displayed as **Y** values in the generated chart.
4. Click **Next**. The Select the fields representing the point location page opens.
5. In the **X value** drop-down list, specify the name of the column of data from your query result set that will be graphed on the x-axis of the scatter or XY chart. Select a data column field from the drop-down list of available data columns in your query result set.
6. Click **Finish**. The wizard closes and the chart is displayed in the editor window.

**Inserting a stock chart:**

Stock charts are used to show fluctuations in stock prices, or other value-based indices, and volumes over time.

**About this task**

To insert a stock chart:

**Procedure**

1. Double-click the **StockChart** object in the **Layouts** palette of the **Palette** view. The StockChart Wizard opens.
2. In the **All Data Sources** tree, navigate to the table or query that will be run to obtain the layout object's query result set and click **Next**. The Select options for a stock chart page opens.

3. In the **Trading interval** drop-down list, select the trading interval over which the stock values are to be charted. Select an interval from the drop-down list of available choices that includes a second, minute, hour, day, week, month or year.

4. In the **Date** drop-down list, specify the name of the data column in your query result set that contains the trading date.

5. In the **High** drop-down list, specify the name of the data column in your query result set that will contain the stocks high price on each date.

6. In the **Low** drop-down list, specify the name of the data column in your query result set that will contain the stock’s low price on each date.

7. Select the **Show open price** check box to display the open price for the stock.

8. In the **Open** drop-down list, select the name of the data column in your query result set that will contain the stock’s open price on each date.

9. Select the **Show close price** check box to display the close price for the stock.

10. In the **Close** drop-down list, select the name of the data column in your query result set that will contain the stock’s close price on each date.

11. Select the **Show volume of trading** check box to display the volume of trading for each stock.

12. In the **Volume** drop-down list, select the name of the data column in your query result set that will contain the volume that the stock traded on each date.

13. Click **Finish**. The StockChart Wizard closes and the chart is displayed in the editor window.

**Inserting a candlestick chart:**

Candlestick charts are used to show gains and losses in stock prices (or other value-based indices) and volumes over time.

**About this task**

To insert a candlestick chart:

**Procedure**

1. Double-click the **CandlestickChart** object in the **Layouts** palette of the **Palette** view. The CandlestickChart Wizard opens.

2. In the **All Data Sources** tree, navigate to the table or query that will be run to obtain the layout object's query result set and click **Next**. The Select options for a candlestick chart page opens.

3. In the **Trading interval** drop-down list, select the trading interval over which the stock values are to be charted. Select an interval from the drop-down list of available choices that includes a second, minute, hour, day, week, month or year.

4. In the **Date** drop-down list, select the name of the data column in your query result set that contains the trading date.

5. In the **Open** drop-down list, select the name of the data column in your query result set that will contain the stock's open price on each date.

6. In the **Close** drop-down list, select the name of the data column in your query result set that will contain the stock's close price on each date.
7. Select the **Show high price** check box to display the high price for the stock.

8. In the **High** drop-down list, select the name of the data column in your query result set that will contain the stocks high price on each date.

9. Select the **Show low price** check box to display the low price for the stock.

10. In the **Low** drop-down list, select the name of the data column in your query result set that will contain the stock’s low price on each date.

11. Select the **Show volume of trading** check box to display the volume of trading for each stock.

12. In the **Volume** drop-down list, select the name of the data column in your query result set that will contain the volume that the stock traded on each date.

13. Click **Finish**. The CandlestickChart Wizard closes and the chart is displayed in the editor window.

**Inserting a timeline chart:**

Timeline charts display a time series as a curve.

**About this task**

To insert a timeline chart:

**Procedure**

1. Double-click the **TimelineChart** object in the **Layouts** palette of the **Palette** view. The TimelineChart Wizard opens.

2. In the **All Data Sources** tree, navigate to the table or query that will be run to obtain the layout object’s query result set and click **Next**. The Select the fields representing the point location page opens.

3. In the **Date** drop-down list, specify the name of the data column from your query result set that will be graphed on the date/time axis of the timeline chart. You should select a data column that includes a date or time value.

4. In the **Value** drop-down list, specify the name of the data column from your query result set that will be graphed on the numeric axis of the timeline chart. You should select a data column that includes numeric values.

5. Click **Finish**. The TimelineChart Wizard closes and the chart is displayed in the editor window.

**Inserting an event band chart:**

Event band charts display a vertical band that represents time intervals for each row returned by a query.

**About this task**

To Insert an event band chart:

**Procedure**

1. Double-click the **EventBandChart** object in the **Layouts** palette of the **Palette** view. The EventBandChart Wizard opens.

2. In the **All Data Sources** tree, navigate to the table or query that will be run to obtain the layout object’s query result set and click **Next**. The Select the fields representing the event time period page opens.
3. In the **Begin date** drop-down list, specify the name of the data column from your query result set that contains the beginning date of the event you are charting in the Event Band chart. You should select a data column that includes a date.

4. In the **End date** drop-down list, specify the name of the data column from your query result set that contains the end date of the event you are charting in the Event Band chart. You should select a data column that includes a date.

5. Click **Finish**. The EventBandChart Wizard closes and the chart is displayed in the editor window.

**Inserting a cluster graph:**

Cluster graphs display data in a recursive and circular set of hubs with spokes.

**About this task**

To insert a cluster graph:

**Procedure**

1. Double-click the **ClusterGraph** object in the **Layouts** palette of the **Palette** view. The ClusterGraph Wizard opens.

2. In the **All Data Sources** tree, navigate to the table or query that will be run to obtain the layout object's query result set and click **Next**. The Select the fields representing the parent-child relationship page opens.

3. From the **Parent field** drop-down list, select the name of the data column in your query result set that will be used for the parent cell in the Cluster graph.

4. From the **Child field** drop-down list, select the name of the data column in your query result set that will be used as the child cell in the Cluster graph.

5. In the **Select the maximum visible child depth** list box, specify the maximum number of child levels that are displayed in the cluster graph under the parent objects for a given zoom level. To see a single level of child levels, type a 1 in the field.

6. Click **Finish**. The ClusterGraph Wizard closes and the graph is displayed in the editor window.

**Inserting a linear map:**

Linear maps display spatial data in a geographic context.

**About this task**

To insert a linear map:

**Procedure**

1. Double-click the **LinearMap** object in the **Layouts** palette of the **Palette** view. The LinearMap Wizard opens.

2. In the **All Data Sources** tree, navigate to the table or query that will be run to obtain the layout object's query result set and click **Next**. The Linear map options page opens.

3. Select **Data contains open boundaries. Plot as Polylines.** to specify that your linear map will be plotted as polylines. Choose this option if the vertices data is
stored in your query result data columns as polylines. When stored as polylines, the vertices have open boundaries. The last point and the first point are different.

4. Select **Data contains closed boundaries. Plot as polygons**. to specify that your linear map will be plotted as polygons. Choose this option if the vertices data is stored in your query result data columns as polygons. When stored as polygons, the vertices have closed boundaries.

5. Click **Next**. The Select the field containing the boundaries page opens.

6. In the **Boundary Field** drop-down list, specify the data column in your query result set that contains the map vertices.

7. Click **Finish**. The LinearMap Wizard closes, and the map displays in the editor window.

**Inserting a SimpleForm object:**

The **SimpleForm** object allows you to optimize data retrieving process when you need to display values from only one row in a query.

**About this task**

You can put several **Label** objects on a **SimpleForm** to display values from different cells of the specified query row.

To insert a SimpleForm:

**Procedure**

1. Double-click the **SimpleForm** object in the **Layouts** palette of the **Palette** view.

2. In the SimpleForm Wizard, navigate to the table or query that will provide data for the **SimpleForm** layout and click **Finish**. The SimpleForm Wizard closes and the **SimpleForm** object is placed on the scene.

3. Add required **Label** objects to the **SimpleForm** layout.
   a. In the **Project Explorer** view, navigate to the **SimpleForm** layout that you inserted.
   b. In the **Palette** view, expand **Primitives** and select **Label**.
   c. Drag the **Label** object to the **Level1** node under **SimpleForm** layout in the **Project Explorer** view.

4. With the help of **Expression Designer**, specify the value to display in each **Label** object.
   a. In the **Project Explorer** view, select the **Label** object you added.
   b. In the **Properties** view, double-click the **Text** property value.
   c. In the Expression Designer window, replace default text with the name of the column that contains the value you want to display. For this, expand **Queries > [Query name] > Output columns** and select the required column.

   **Note:** **[Query name]** stands for the name of the query or table that you specified for the **SimpleForm** layout.

5. Click **OK** to close the Expression Designer window.

**Results**

The **SimpleForm** layout with all required labels is added to the scene. You can check the result in Runtime mode.
Inserting a matrix:

Matrices display data in a two-dimensional array of cells.

About this task

To insert a matrix:

Procedure

1. Double-click the created object and add any controls (for example, labels) with the necessary values to display them in the [object_name].
2. In the All Data Sources tree, navigate to the table or query that will be run to obtain the layout object's query result set and click Next. The Specify matrix options page opens.
3. Specify the width of each cell in the Cell width field.
4. Specify the Height of each cell in the Cell height field.
5. Specify the gutter width (space between cells horizontally) in the Gutter width field.
6. Specify the gutter height (space between cells vertically) in the Gutter height field.
7. Specify the number of columns that will be displayed in the matrix in the Number of columns field.
8. Select the Display cell border check box to display a border around each cell in the matrix.
9. Click Finish. The Matrix Wizard closes and the matrix is displayed in the editor window.

Inserting a spiral:

The Spiral object displays data in an infinite inward spiral, in which each data point becomes smaller.

About this task

The Spiral layout is useful for displaying data that is not mathematically related. Zooming in on the Spiral object reveals more data further into the spiral, and zooming out shows more data on the outer portion of the spiral.

To insert a spiral:

Procedure

1. Double-click the Spiral object in the Layouts palette of the Palette view.
2. In the Spiral Wizard, navigate to the table or query that will provide data for the Spiral object.
3. Click Finish. The Spiral Wizard closes and the Spiral object is displayed on the scene.

Results

You can customize the look and behavior of a Spiral object by modifying its properties and properties of its supporting objects: DataTemplate and Ellipse.
Working with nodes of TreeChart and OrganizationChart:

You can insert TreeChart or OrganizationChart into a visual dashboard and expand or collapse its nodes.

Inserting an organization chart:

Organization charts display data in a tree structure that reads from top to bottom and is typically associated with a personal hierarchy.

About this task

To insert an organization chart:

Procedure

1. Double-click the created object and add any controls (for example, labels) with the necessary values to display them in the [object_name].
2. In the All Data Sources tree, navigate to the table or query that will be run to obtain the layout object’s query result set and click Next. The Organization chart options page opens.
3. In the Parent field drop-down list, select the name of the data column in your query result set that will be used for the parent cell in the Organization chart.
4. In the Child field drop-down list, select the name of the data column in your query result set that will be used as the child cell in the Organization chart.
5. Select the Include a label check box to include a label in each of the parent and child cells of the Organization chart.
6. In the Label field drop-down list, specify the name of a data column in your query result set that will contain the label information for each parent and child cell in the organization chart. You must select a data column from the drop-down list of available data columns that are in your query result set if you selected the Include a label check box.
7. In the Size of child objects relative to parent objects field, specify the size of the child objects as they are displayed in the Organization chart relative to the size of the parent objects. Type a number between .01 and 1 in the field.
8. In the Layout Model drop-down list, select the layout view option either classic or horizontal.
9. Click Finish. The OrganizationChart Wizard closes and the chart is displayed in the editor window.

Inserting a tree chart:

Tree charts display data in a tree structure showing a cascading set of parent-child relationships.

About this task

To insert a tree chart:

Procedure

1. Double-click the created object and add any controls (for example, labels) with the necessary values to display them in the [object_name].
2. In the All Data Sources tree, navigate to the table or query that will be run to obtain the layout object's query result set and click Next. The Tree chart options page opens.

3. In the Parent field drop-down list, select the name of the data column in your query result set that will be used for the parent cell in the Tree chart.

4. In the Child field drop-down list, select the name of the data column in your query result set that will be used as the child cell in the Tree chart.

5. Select the Include a label check box to include a label in each of the parent and child cells of the Tree chart.

6. In the Label field drop-down list, select the name of a data column in your query result set that will contain the label information for each parent and child cell in the tree chart. You must select a data column from the drop-down list of available data columns that are in your query result set if you selected the Include a label check box.

7. In the Number of levels to initially display field, specify the maximum number of levels that are displayed initially in the tree chart under the parent objects. Type the number of levels in the field. A value of 1 will display one level.

8. Click Finish. The TreeChart Wizard closes and the chart is displayed in the editor window.

Modifying the nodes state:

You can modify the state of the TreeChart or OrganizationChart nodes on a visual dashboard.

Inserting an example chart:

See the following example of adding an OrganizationChart to a visual dashboard. This visual project is created on the basis of DEMO.EMPLOYEE table from the default Samples repository.

About this task

To create a task on the basis of DEMO.EMPLOYEE table:

Procedure

1. Create a visual dashboard.

2. Insert the query with the following SQL statement:
   ```sql
   SELECT A.EMPLOYEE_ID, A.FULL_NAME, A.SUPERVISOR_ID
   FROM DEMO.EMPLOYEE A
   ```

3. Add OrganizationChart to the visual project.

4. On the second page of the OrganizationChart wizard specify the following chart options:
   a. Set SUPERVISOR_ID in the Parent field.
   b. Set EMPLOYEE_ID in the Child field.
   c. Set FULL_NAME in the Label field.

5. Click Finish. The example chart is added to the current visual project.
Saving the chart nodes state:

You can save the chart nodes state in the TreeChart or OrganizationChart on a visual dashboard programmatically. See an example of saving the chart nodes state on the basis of the DEMO.EMPLOYEE example project.

Before you begin

You must have a visual dashboard with an added OrganizationChart or TreeChart.

About this task

To save the chart nodes state:

Procedure
1. In the DEMO.EMPLOYEE example project, add two global parameters ExpandedNodesSaved and CollapsedNodesSaved of the Text type.
2. Add two buttons to the project: Button1 and Button2.
3. For Button1 specify the Caption property as Save.
4. Add the action Set Values on the Click event. In the Add New Action wizard specify two actions:
   b. CollapsedNodesSaved=OrganizationChart1.DataLocator.CollapsedNodes
5. For Button2 specify the Caption property as Load.
6. Add the action Set Values on the Click event. In the Add New Action wizard specify two actions:
   b. OrganizationChart1.DataLocator.CollapsedNodes=CollapsedNodesSaved

   Note: At first, it is necessary to bind the value of ExpandedNodes, and then the value of the CollapsedNodes property.
7. Click Runtime tab. Expand and collapse the nodes. Click the Save button to write the nodes state to the global parameters.
8. Expand and collapse the nodes, click the Load button to restore the tree to the state saved by clicking the Save button.

Expanding or collapsing all chart nodes:

You can expand or collapse all chart nodes in the tree or organization charts on a visual dashboard programmatically. See an example of expanding or collapsing the chart nodes on the basis of the DEMO.EMPLOYEE example project.

Before you begin

You must have a visual dashboard with an added OrganizationChart or TreeChart.

About this task

To expand or collapse all the chart nodes:

Procedure
1. In the DEMO.EMPLOYEE example project, add a button and set its Caption value to Expand All.
2. Add the action **Set Values** on the **Click** event with the expression:
   a. `OrganizationChart1.DataLocator.ExpandedNodes=field("Query1", "EMPLOYEE_ID")`

   **Note:** The separator in the expression depends on the locale settings.

3. Add a button and set its **Caption** value to **Collapse All**.

4. Add the action **Set Values** on the **Click** event with the expression:
   a. `OrganizationChart1.DataLocator.CollapsedNodes=field("Query1", "EMPLOYEE_ID")`

   **Note:** The separator in the expression depends on the locale settings.

5. Click **Runtime** tab and expand all chart nodes by clicking the **Expand All** and collapse them all by clicking **Collapse All**.

   **Note:** If you add the example buttons to the DataTemplate they will work only with the node connected with this DataTemplate.

---

**Expanding or collapsing all chart nodes by selecting their labels:**

You can expand or collapse the current chart node in the tree or organization chart on a visual dashboard by selecting their labels. See an example of expanding or collapsing the chart nodes on the basis of the DEMO.EMPLOYEE example project.

**Before you begin**

You must have a visual dashboard with an added OrganizationChart or TreeChart.

**About this task**

To expand or collapse a chart node:

**Procedure**

1. In the DEMO.EMPLOYEE example project, in the **Project Explorer** view expand the **OrganizationChart1** component, **DataTemplate1(Query1)**, **Level1** and select **Label1**.

2. Add the **Set Values** action for the **Click** event of the **Label1**:
   
   ```
   ```

3. Click **Runtime** tab and expand or collapse chart nodes by clicking their labels.

**Expanding to a particular chart node:**

You can open a particular chart node in the tree or organization chart on a visual dashboard programmatically. See an example of expanding or collapsing the chart nodes on the basis of the DEMO.EMPLOYEE example project.

**Before you begin**

You must have a visual dashboard with an added OrganizationChart or TreeChart.

**About this task**

To open a particular chart node in the tree or organization chart:
Procedure
1. In the DEMO.EMPLOYEE example project, add the Combo object.
2. In the Combo wizard, specify the Query1 that is used for the OrganizationChart1.
3. Add EMPLOYEE_ID column values to the Combo.
4. Expand OrganizationChart1 component, DataTemplate1(Query1), Leve11, and then select Label1.
5. Modify its Frame.FillStyle.FillColor property by adding an expression:
   =if(EMPLOYEE_ID=Combo1.Selection,#F00000,#78b1ea). To highlight the node after selecting it.
   
   **Note:** The separator in the expression depends on the locale settings.
6. Add the Set Values action for the Change event of the Combo object:
   
   ```
   OrganizationChart1.DataLocator.CollapsedNodes=field("Query1","EMPLOYEE_ID")
   OrganizationChart1.DataLocator.ExpandedNodes=Combo1.Selection
   OrganizationChart1.DataLocator.CollapsedNodes=Combo1.Selection
   ```
7. Click Runtime tab. Select the required IDs from the Combo object, the proper nodes will be expanded and highlighted with red.

Inserting a dial:

Dials display points of data as a range between two numbers on a metered scale.

About this task

To insert a dial:

Procedure
1. Double-click the Dial object in the Layouts palette of the Palette view. The Dial Wizard opens.
2. Select the minimum value that is displayed on the dial in the Min list.
3. Select the maximum value that is displayed on the dial in the Max list.
4. Select the starting angle of the dial in the Start list. For example, specifying -90 will display a horizontal line that stretches from the center of the dial to the left of the canvas.
5. Select the ending angle of the dial in the End list. For example, specifying 90 will display a horizontal line that stretches from the center of the dial to the right of the canvas.
6. Select the Symmetric check box to automatically make the dial symmetrical. This option sets the End angle as opposite to the Start angle. For example, if the Start angle is set to -110, the End angle is automatically set to 110.
7. Select Label Inside to display the major tick mark labels in the inside of the arc of the dial.
8. Select Label Outside to display the major tick mark labels on the outside of the arc of the dial.
9. To add a sector to the dial, specify a value in the Sectors field and click the Add New Entry button. The Max value is automatically changed to reflect the sum of the specified sectors. For example, as a default, there are three sectors set at 33.33 each. This creates a dial with three sectors, each a third of the total dial. If you add another sector with a value of 20, the Max value is set to 120.
10. To delete a sector, highlight an existing sector in the list and click the **Delete Entry** button.

11. Click **Finish**. The Dial Wizard closes and the new dial object is displayed on the dashboard canvas.

**Inserting a solid chart:**

Solid charts display your query results data in a visually rich format, like charts for all query types in DataQuant. In contrast to other composite charts that are available in visual projects, solid charts are added as the whole object to a scene. Solid charts are fully customizable.

**About this task**

To insert a solid chart object:

**Procedure**

1. Double-click the **SolidChart** object in the **Layouts** palette of the Palette view. The SolidChart Wizard opens.
2. Select a query for which you want to create a chart and click **Next**.
3. If the selected query has a saved chart, you can add it to your visual project on the **Chart** page, and click **Finish** to close the wizard.
4. To create a chart, on the **Chart type** page, specify the type of the chart. For more information about chart types, see [Specifying chart type options](#) topic. Click **Next**.
5. On the **Data series** page, specify the columns of the result set that you want to use to generate data points of a chart. For more information, see [Specifying data series options](#) topic. Click **Next**.
6. On the **Format** page, specify the options to define the visual output of the chart. For more information, see [Specifying format options](#) topic.
7. Click **Finish** to close the SolidChart Wizard. The created chart is displayed on the scene.

**Editing a solid chart:**

You can edit the added solid chart to customize it for your particular purposes.

**About this task**

To edit a solid chart:

**Procedure**

1. In **Canvas** mode, right-click the solid chart that you want to edit and select **Edit Chart**.
2. In the "Edit chart properties" window, you can edit the following options:
   - Chart type
   - Data series
   - Chart format

For detailed information about editing the solid chart options see the topics:

- [Specifying chart type options](#)
- [Specifying data series options](#)
- [Specifying format options](#)
3. When you finish editing the solid chart options, click **OK** to apply your changes.

**Changing a layout object's query:**

You can change a layout object's query by selecting **Change Query** in the **Project Explorer** menu.

**About this task**

To change a layout's object's query:

**Procedure**
1. In the **Project Explorer**, right-click the data template folder for the layout object whose query you want to change.
2. Select **Change Query** from the context menu. The Select Query window opens where you can select a query that has been added to your **Queries** folder.
3. Click **OK**. The Layout object will use the new query that you selected.

**Specifying a query parameter value:**

You can specify query parameter values for layout objects in the **Parameters** folder of the layout object's data template.

**About this task**

To specify a query parameter value:

**Procedure**
1. From the **Project Explorer** view, expand the data template folder for the layout object whose query parameter you want to set. Expand the **Parameters** folder.
2. Select the parameter from the **Parameters** folder. The name of the parameter and the current default value for the parameter are displayed in the **Properties** view.
3. Specify a new value for the parameter in the **Properties** view. You can specify a literal or an expression that will be evaluated at runtime.

**Extending a layout object's functionality:**

Layout objects are the most versatile objects that you can insert in a visual project. As originally created by the layout wizard, the layout object is a simple, one-dimensional representation of a single set of query results data. However, as you add functionality, a single layout object can become a dynamic, multidimensional representation of data obtained from more than a single query.

**About extended layout objects:**

Extended layout objects allow you to roll up data across one or more columns in a result set.

Extended layout objects are different from standard layout objects in that they allow for a visual, client-side OLAP roll up of data across one or more columns in the associated query result set. This enables visual designers to create visual representations of data that are more organized and easy to use. Users can drill down into specific data symbols to obtain only the data they need, without having
to view the entire result set at once. For example, the Q.STAFF table included in the sample repository has thirty-five rows, one for each employee in the company. A pie chart displaying the salary of all thirty-five employees would be visually cluttered and confusing. Rather than do this, the Extended option allows you to roll up salary data across one or more columns in the result set, providing on-demand drilldown to the underlying data. In the case of the Q.STAFF table, we can elect to roll the salary across department, followed by job type. Under this mode, the pie chart will display a slice for each department, with plus signs that let you expand each wedge to display each job type in that specific department. Expanding each job type slice then presents individual salary information for the specific department and specific job type. You can add as many drilldown operations to your layout objects as you want by specifying one or more columns to be moved to the side group in the query results or layout wizard.

Displaying OLAP data in layout objects:

Displaying OLAP data in layout objects allows you to organize visual data into manageable hierarchies.

Certain layout objects allow you to display OLAP data. The layout objects that allow you to display OLAP data are:

- Bar Charts
- Column Charts
- Pie Charts

You can display OLAP data in these layouts by selecting an OLAP query from a multidimensional data source. The benefit of this is that it allows visual designers to choose along which dimensions they would like data to be displayed, and allows users to drill down into specific levels of detail. By progressively expanding portions of levels within a dimension, it is possible to drill-down into progressively more detail. At each level of detail, the layout object's data symbols refresh to visually represent that level's data points.

For example, you have a cube model that contains dimensions for product data, regional data, and time data. The product dimension contains individual products that are organized into four product families. The region dimension contains states that are organized into four geographic regions. The time dimension contains the four fiscal quarters of the current year. You want to create a column chart that displays profit totals for all products, and organize the visual data by time and region. You set the column chart to display data points from the PROFIT column as bars, and select the Time and Region dimensions to organize the data. The column chart is initially displayed as four bars that show the profit totals for each region for the whole year. Expandable cells that display the current dimensional levels are displayed under each column. The user can drill down into any region, and the column chart refreshes to display a column for every state in that region, while still displaying the other three regional totals. The user can then drill down further to display the quarterly totals for a specific state. Alternatively, the user can choose to drill down into a given region's year, displaying that region's quarterly totals. This dynamic organization allows users to quickly and easily make visual comparisons of data as it pertains to the larger data set.

Another way to organize OLAP data in a layout object is to use slicer objects. A slicer is used to filter the multidimensional data that is displayed in a layout object. A visual designer creates a slicer and selects a dimension of the layout object's OLAP query to filter. When the user clicks the slicer object in runtime, a window opens that lists all of the selected dimension's hierarchies and hierarchy...
elements. The user can then check which elements the layout object will display. Whenever the user changes the settings in the filter window, the layout object refreshes to reflect the change in filtered data.

For example, in the layout object mentioned above, the data is organized by time and region and always displays the product totals for each data symbol. You want to give users the option to select which particular product totals or product group totals are displayed at any given time. You insert a slicer object, select the Product dimension to filter the data, and connect the slicer object to the layout object using the **Connectivity** tool. When the user clicks the slicer object in runtime, a window opens that displays a tree containing all of the product dimension’s hierarchy elements. When the user checks off the products and clicks **OK**, the layout object automatically refreshes to display only the totals for the selected products.

*Changing the default data symbol:*

Each graphical layout object includes a default data symbol object which is used within the layout object to display each row of the query results data as a data point. For most layout objects, you can change the default data symbol and modify its properties.

**About this task**

To change a layout object's default data symbol:

**Procedure**

1. Double-click the layout to display the contents of its data template.
2. Select the existing data symbol and press **DELETE**.
3. Select the new data symbol that you want to use from the **Palette** view and insert it in the editor. The name of the new data symbol is added to the data template folder in the **Project Explorer** view.
4. You can use the **Properties** view to modify the properties of the data symbol, such as its shape or color.

*Changing the default connector:*

Many graphical layout objects include a default connector object which is used within the layout to connect each data point. For those layout objects that include connectors, you can change the default connector and modify its properties.

**About this task**

To change a layout object's default connector:

**Procedure**

1. From the **Project Explorer**, select the default connector name from the data template folder and press **Delete**.
2. Select the new connector that you want to use from the **Palette** view and insert in the editor. The layout object that includes the new connector is presented in the editor window. The name of the new connector is added to the data template folder in the **Project Explorer**.
3. You can use the **Properties** view to modify the properties of the connector.
Displaying multiple graphic objects for each data point:

For most layout objects you can add additional graphics that will be displayed with each instance of the data symbol for each row of data that is included in the query result set.

About this task

For example, the default data symbol in a pie chart is a wedge. You can specify that each wedge of the pie chart additionally displays with a text label and two radio buttons. You can choose to add any of the graphic objects that are included in the Palette view with a few exceptions.

To add graphic objects for display with each row of query data:

Procedure
1. Double-click the layout object to display the contents of its data template.
2. Select the layout object that you want to add from the Palette view and insert it in the editor window. The new graphic object is drawn in the editor window. The name of the graphic object is added to the data template folder in the Project Explorer.
3. You can use the Properties view to modify the properties of the data symbol, such as its shape or color

Displaying multiple query results in a single layout:

For many layout objects, you can display the results of multiple queries in a single layout object.

About this task

For example, you could create a single XY chart that displays sales figures derived from one query and spending figures derived from another query. Each set of query results data that is drawn in a single layout object has its own data template. When you include multiple data templates (sets of query results) in a layout, each data template is drawn in the order in which it appears in the Layout folder in the Project Explorer view. The data template that appears last in the folder is drawn last and therefore appears on top of all previous layers.

You can add data templates to the following layout objects:
- CandlestickChart
- EventBandChart
- LinearMap
- ScatterChart
- StockChart
- TimelineChart
- ColumnChart
- BarChart
Copying an existing data template:

You can add a data template to a layout object by copying an existing data template and then changing the query that is associated with the original data template.

About this task

When you copy data templates, all child objects in the data template folder are also copied. As a result, you get an exact duplicate of the existing data template. You would then change the query that is associated with the new data template and you can optionally modify any child object properties.

To add a data template to a layout object by copying an existing data template:

Procedure

1. In the Project Explorer view, right-click the data template folder that you want to copy. Select Copy from the pop-up menu. The data template folder is copied to the clipboard.
2. In the Project Explorer view, right-click the layout object folder to which you want to add the data template. Select Paste from the pop-up menu. The data template folder is copied to the layout object’s folder.
3. You can change the query that is associated with the data template in one of the following ways:
   - Right-click the new data template folder that you just added and select Change Query. The Select Query window opens where you can select a query that has been added to your Queries folder.
   - If the data template that you copied uses a query that has parameters, you can change the query for the data template by changing the value of the query parameter. Expand the data template folder that you just added and select the Query Parameters folder. Set the query parameter to a new value.
4. After a data template is added to the layout object, you can modify the objects and their properties as you require.
5. Click Runtime in the editor to view the results.

Creating a new data template:

You can use the Insert Data template command to add a new data template to a layout object.

About this task

When you add a new data template to a layout object you have to add the graphic objects that will be used to display the data in that layer.

To add a new data template to a layout object:

Procedure

1. In the Project Explorer view, right-click the layout object folder to which you want to add the data template. Select New > Data Template from the pop-up menu. The Data Template wizard opens.
2. From the Data Template wizard select a query that will be used to obtain the data for this data template. The queries that are listed in the Data Template wizard are the ones that you have added to your Queries folder.
3. After a data template is added to the layout, you can modify the objects and their properties as you require. Select the data template folder that you added. A new canvas for the data template is opened in the editor window.

4. Click **Runtime** in the editor to view the results.

**About data templates:**

Data templates visually represent query result set data in layout objects.

Data templates are used to create and control the visual representation of query result set data within layout objects. Anything that you place within a data template is replicated for each row of data that is returned from the query associated with the template. For example, the template for a line chart can consist of a data symbol and a connecting line. Each row of query data will then be represented by its own data symbol and connecting line. With direct access to the data template, content developers have the ability to construct a wide variety of customized charts and layouts. For example, one could produce a floating quartile graph by setting the start and end points of each bar in the data template and adding horizontal line primitives that are set to the mean and 25th and 75th percentiles of the data column(s).

Data templates provide tremendous control over a layout’s behavior. However, you can also use them to make minor modifications to your charts and layouts. The following paragraphs provide a number of typical modifications that you can want to make to your data templates.

**Customizing data symbols and graphic objects**

You can customize the way query results are displayed in layout objects by editing the properties of the objects contained in their data templates. For example, you want to build an organization chart that displays the name of each member of a sales team in bold. By default, the text of the label objects that display each name is not bold. To display each name with bold text, you change the Bold property of the label object in the organization chart’s data template folder to true. Now each member of the sales team’s name will be displayed with bold text. You can also parameterize the display properties of data symbols and graphic objects. You want the names of sales team members who met their quotas to display as black text, and the names of sales team members who did not meet their quotas to display as red text. You change the Color property value of the label objects in the data template to a conditional expression. Now, when the organization chart is run, each name will display as either red or black depending on whether the sales team member met his or her quota.

**Displaying multiple graphic objects for each data point**

You can display multiple objects for each data point by adding multiple objects to the data template folder of the layout object. Using the example of an organization chart again, you might want to add a column chart that shows sales team member’s quarterly sales totals by their name. You open the data template folder in the editor window and add a parameterized ColumnChart object so that it is positioned directly beneath the label object of the organization chart. Now when the organization chart is run, a column chart is displayed for each team member that shows their individual quarterly sales totals. Now when the organization chart is run, a column chart is displayed for each team member that shows their individual quarterly sales totals.
Displaying multiple query result sets in a single layout object

A data template is always associated with a single query. However, you can draw from as many columns as you want when presenting the data in your data template. For example, you want to chart three query columns and set the color or size of each data point, based on three other query columns, and so on.

A layout can have multiple data templates, each of which draws data from differing queries, running against the same or differing data sources. When you add multiple data templates to a layout, such as a line chart, all data points are presented within the same pair of axes, auto-scaled (if set) to the minimum and maximum data values across all queries/data templates. Finally, you can mix the visual representation in a given layout. For example, you can present one or more line chart traces within a column chart, each running from the same or differing data templates.

Modifying a data template’s visibility:

You can specify whether a data template layer is displayed or not displayed in the layout object.

About this task

The visibility property for a data template is a Boolean value that you can set to true or false. You can also enter a property expression that returns a true or false value.

To specify the visibility of a data template within a layout object:

Procedure

1. From the Project Explorer view, select the data template folder whose visibility property you would like to change. All of the properties for that data template are displayed in the Properties view.
2. Select True or False from the Visibility property list box or specify a property expression that returns True or False.

Results

Note: For layout objects that do not have a data template folder, visibility is a property of the layout object. You would select that layout object, and modify its Visibility property.

Nesting layout objects:

When you nest layout objects, you can pass query result information from a higher level object to a lower level object. Because you have the ability to pass this information, you can use nested layouts to display more detailed information that relates to a specific data value.

About this task

For example, for each box in an employee Organization chart, you could also display a bar chart with the salary history for that particular employee.

To nest one layout object within another layout object:
Procedure
1. Double-click the layout to display the contents of its data template.
2. The canvas for that data template folder opens in the editor window.
3. Select the new layout object from the Palette view. Insert the new layout object in the editor window. The [layout name] Wizard opens. Specify the name of the query that will be used to obtain the data for this child layout. The queries that are listed in the [layout name] Wizard are the ones that you have added to your Queries folder for the visual project.
4. When you are creating the query that will be used by a child layout object you can do one of the following:
   • Create a query that references the parent layout object's query results using a query parameter. By referencing the parent query results, you can specify how to display your child layout.
   • Create a completely new query that does not reference the parent layout object's query results. If you do not reference the parent object query results, then your child object's complete query results set will display for every row of the parent object's query result set.
5. After the new layout object is inserted, you can modify its properties as you require.
6. Click Runtime in the editor to view the results.

Working with map objects
You can display Google Maps and add different objects to them in your visual dashboards.

Before you begin

Note: Google Maps can be rendered only in HTML5 mode.

About this task

The following Google Map objects are available:
• GoogleMap
• GoogleMapMarker
• GoogleMapPolyline
• GoogleMapPolygon
• GoogleMapObject

Inserting a Google Map object:

You can use the GoogleMap object to display Google Maps with different objects on them in your visual dashboards.

About this task

To insert a Google Map object:

Procedure
1. In the Palette view, expand the Map palette and double-click the GoogleMap object to open the GoogleMap Wizard.
2. In the Viewpoints area, specify the start position of the GoogleMap object and its zoom level.
• Select **Auto** check box if you want to set a map viewpoint automatically depending on the location of the objects that you added to the map.

• Clear **Auto** check box and enter the coordinates in the **Latitude** and **Longitude** fields to specify the map center.

• From the **Zoom level** list, select the zoom value for the **GoogleMap** object.

• Select **Fixed** check box if you want to fix the map viewpoint. In this case, the users are not able to scroll a map or zoom it.

  **Note:** When you select the **Fixed** check box, the Position, Zoom, Navigation, and Overview Map controls becomes unavailable in the **GoogleMap controls** area.

3. In the **GoogleMap controls** area, select the controls that you want to add to the **GoogleMap** object and click **Next**.

4. In the **Available map types** area, select the map types that are available for the users.

5. From the **Map type by default** list, select the map type that is loaded by default and click **Next**.

6. Define the content that you want to add to the **GoogleMap** object and click **Next**.

   • Select **Create a map with a component** if you want to add an object to a map to display specific data, and then select the appropriate object from the **Select Component** area.

   • Select **Create an empty map** if you want to add a map without any objects on it.

7. Depending on the option that you selected, specify the required information on the next pages of the GoogleMap Wizard and click **Finish** to display the **GoogleMap** object that you created in the editor.

  **Note:** To open the **GoogleMap** object that you created in DataQuant for WebSphere, you must obtain and specify the correct Google Map API Key.

**Inserting a GoogleMapMarker object:**

You can use the **GoogleMapMarker** object to add markers to the **GoogleMap** object by specifying the coordinates of the markers.

**Before you begin**

The **GoogleMapMarker** object properties are defined in the **User’s reference** section of the DataQuant help.

**About this task**

The **GoogleMapMarkers** objects are interactive points on a map.

To add markers to the **GoogleMap** object:

**Procedure**

1. When you insert the **GoogleMap** object, select **Create a map with a component**, and then select the **Marker** option on the **Define map content** page of the GoogleMap Wizard. Then click **Next**.

2. Select **No** to enter the coordinates of the markers manually and click **Next**.

3. Define the markers on the **Define the properties for the markers** page.
• In the Caption column, enter the name that you want to use for a marker.
• In the Tooltip column, you can specify the text of a tooltip for a marker.
• In the Latitude column, specify the latitude coordinate.
• In the Longitude column, specify the longitude coordinate.

4. When you finish specifying the information for the markers, click Finish to close the GoogleMap Wizard and display the GoogleMap object with the markers that you added to it in the editor.

Tip: To add any other objects to the GoogleMap object that you created, drag them from the Map palette to the GoogleMap object in the Project Explorer view and specify the object properties.

Inserting a GoogleMapMarker object to display query results:

You can use the GoogleMapMarker object to display query results as geographic markers on the GoogleMap object.

Before you begin

The GoogleMapMarker object properties are defined in the User's reference section of the DataQuant help.

About this task

The GoogleMapMarkers objects are interactive points on a map.

To display query results as markers:

Procedure

1. When you insert the GoogleMap object, select Create a map with a component, and then select the Marker option on the Define map content page of the GoogleMap Wizard. Then click Next.
2. Select Yes to display query results as geographic markers on the GoogleMap object and click Next.
3. In the All Data Sources tree, select a table or query, which results you want to use to define markers on the map, and click Next.
4. From the Latitude field list, select a column to supply the latitude coordinate for the map markers.
5. From the Longitude field list, select a column to supply the longitude coordinate for the map markers.
6. From the Caption field list, select a column to populate the labels for the map markers.
7. Select the Enable information window check box to allow the users to open an information window when they click the markers on the map.
8. From the Information field list, select a column to populate the information windows of the map markers.
9. Click Finish to close the GoogleMap Wizard and display the GoogleMap object with the markers that you added to it in the editor.

Tip: To add any other objects to the GoogleMap object that you created, add a new Data Template to it, and then drag the required object from the Map pallet to the Data Template in the Project Explorer view.
Inserting a GoogleMapPolyline object to display query results:

You can use the GoogleMapPolyline object to display query results as a geographic map polyline.

Before you begin

The GoogleMapPolyline object properties are defined in the User’s reference section of the DataQuant help.

About this task

The GoogleMapPolyline object creates a linear overlay on Google Maps. It consists of a series of points and creates a series of line segments that connect the points in the ordered sequence.

To insert a GoogleMapPolyline object:

Procedure

1. When you insert the GoogleMap object, select Create a map with a component, and then select the Polyline option on the Define map content page of the GoogleMap Wizard. Then click Next.
2. In the All Data Sources tree, select a table or query, which results you want to use to define a polyline on the map, and click Next.
3. Select the Direct check box to define the latitude and longitude coordinates of the polyline points and click Next.
   a. From the Tooltip field list, select a column to supply a tooltip text for the polyline on the map.
   b. From the Vertices field list, select a column to supply the coordinates for the polyline points.
   c. From the Point format list, select the format for coordinates (Latitude-Longitude or Longitude-Latitude).
4. Select the Encoded check box to define a polyline in the Google Maps encoded format and click Next.
   a. From the Tooltip field list, select a column to supply a tooltip text for the polyline on the map.
   b. Select the columns to specify a series of points within a polyline from the Points field, Levels field, NumLevels field, ZoomFactor field lists.
5. Click Finish to close the GoogleMap Wizard and display the GoogleMap object with the polyline that you added to it in the editor.

Tip: To add any other objects to the GoogleMap object that you created, add a new Data Template to it, and then drag the required object from the Map pallet to the Data Template in the Project Explorer view.

Inserting a GoogleMapPolygon object to display query results:

You can use the GoogleMapPolygon object in your visual dashboard to display query results as a geographic map polygon.

Before you begin

The GoogleMapPolygon object properties are defined in the User’s reference section of the DataQuant help.
About this task

Like the GoogleMapPolyline object, the GoogleMapPolygon object represents a series of sequential data points. However, instead of being open-ended, polygons are designed to define regions within a closed loop.

You can define the characteristics of the polygon edge and the fill area within the enclosed region.

To insert a GoogleMapPolygon object:

Procedure
1. When you insert the GoogleMap object, select Create a map with a component, and then select the Polygon option on the Define map content page of the GoogleMap Wizard. Then click Next.
2. In the All Data Sources tree, select a table or query, which results you want to use to define a polygon on the map, and click Next.
3. Select the Direct check box to define the latitude and longitude coordinates of the polygon points and click Next.
   a. From the Tooltip field list, select a column to supply a tooltip text for the polygon on the map.
   b. From the Vertices field list, select a column to supply the coordinates for the polygon points.
   c. From the Point format list, select the format for coordinates (Latitude-Longitude or Longitude-Latitude).
4. Select the Encoded check box to define a polygon in Google Maps encoded format and click Next.
   a. From the Tooltip field list, select a column to supply a tooltip text for the polygon on the map.
   b. Select the columns to specify a series of points for a polygon from the Points field, Levels field, NumLevels field, ZoomFactor field lists.
5. Click Finish to close the GoogleMap Wizard and display the GoogleMap object with the polygon that you added to it in the editor.

Tip: To add any other objects to the GoogleMap object that you created, add a new Data Template to it, and then drag the required object from the Map pallet to the Data Template in the Project Explorer view.

Inserting a GoogleMapObject object to display query results:

You can use the GoogleMapObject object to display query results as a geographic map object.

Before you begin

The GoogleMapObject properties are defined in the User’s reference section of the DataQuant help.

About this task

To display query results as a geographic map object:
Procedure

1. When you insert the GoogleMap object, select Create a map with a component, and then select the Object option on the Define map content page of the GoogleMap Wizard. Then click Next.

2. In the All Data Sources tree, select a table or query which results you want to use to define the object on the map.

3. From the Tooltip field list, select a column to supply the text of a tooltip for the object.

4. From the Bounds field list, select a column to supply the boundaries of the object on the map.

5. From the Content field list, select a column with the content that you want to display on the map, for example an image.

6. Click Finish to close the GoogleMap Wizard and display the GoogleMap object with the object that you added to it in the editor.

Tip: To add any other objects to the GoogleMap object that you created, add a new Data Template to it, and then drag the required object from the Map pallet to the Data Template in the Project Explorer view.

Working with the Layers view

The Layers view allows you to group palette objects into separate layers that can be displayed or hidden during design time.

About this task

Layers aid the design process by allowing you to display and hide selected objects that overlap each other on a scene’s canvas. Layers are grouped together within each level of detail of a scene, and each palette object is initially placed in the Default layer. Any number of layers can be added to each level of detail. To open the Layers view:

Procedure

1. Open a visual dashboard and select a level of detail of a scene.

2. Select Window > Show View > Layers. The Layers view opens and the current level of detail’s layers are displayed.

3. To add a layer to the current level of detail, right-click the level in the Layers view and select Insert Layer. A new layer is added to the tree.

4. To rename a layer, right-click the layer, select Rename Layer [Layer Name], and specify a new name for the layer.

5. To delete a layer, right-click the layer and select Delete. The layer is deleted.

6. To add a palette object to an existing layer, right-click the palette object on the scene canvas, select Layer, and select the layer from the pop-up menu. The palette object is added to the specified layer in the Layers view.

7. To move a palette object to a different layer, in the Layers view, click and drag the object to the layer.

8. To hide a layer’s contents, clear individual objects or the layer itself in the Layers view. If a layer is cleared, all of the objects that are assigned to the layer are hidden on the scene canvas. If an individual object is cleared, only that object will be hidden.
Working with global resources

Globals are resources that can be used by all the pages of a visual report or all the scenes of a visual dashboard.

About this task

You can use globals for computing object properties. All global resources are displayed in the **Globals** folder of the **Project Explorer**. You can choose to have one or more of the following globals available to the entire visual project:

- **Parameters**
  Parameters are available to your entire visual project and enable you to use variable values when specifying object properties.

- **Color Maps**
  A color map can be used to vary the color of a visual project object depending on a value associated with the object.

- **Color Sequences**
  A color sequence can be used to modify the color properties of a visual project object based on an index value.

- **Security lists**
  Security lists can be used to tailor visual report or dashboard content to the security level of the user who is viewing the report or dashboard.

- **Images**
  A stock image enables you to store your image resources with your visual project. Storing image resources with your visual project increases the portability and performance of the project.

- **Expiration Schedules**
  Expiration schedules can be set up to dictate when given data expires.

- **Java Script Modules**
  Java Script Modules enable you to use custom JavaScript functions in visual projects.

- **Translation Tables**
  Translation tables enable dashboard designers to publish a single visual dashboard that can be viewed in multiple languages.

Working with color sequences

A color sequence is a type of global resource that can be used to modify the color properties of a visual project object based on an index value. Each index value specifies which wedge in a color sequence wheel will contain the color for an object.

About this task

You are given a default color sequence that includes set colors in the color sequence wheel. You can also set up unique color sequences that will vary the colors that are included in the color sequence wheel. You specify that a visual project object will use a color sequence when you are specifying the properties of that object. At that time you also decide whether you will specify the index value for a color sequence with an explicit number, an expression or a function.
Creating a new color sequence:

You use the New Color Sequence wizard to create a color sequence for your visual project.

About this task

To create a new color sequence:

Procedure

1. You open the New Color Sequence wizard in one of the following ways:
   - From the Project Explorer view, click the Globals node. Right-click the Color Sequences node. Select Insert Color Sequence from the pop-up menu. The New Color Sequence wizard opens.
   - From the Project Explorer view, click the Globals node. Double-click the Color Sequences node. The New Color Sequence wizard opens.
2. Type a unique name for the color sequence in the Name field.
3. The colors that will appear in the color sequence are displayed in the Colors wheel. Clicking on each wedge will display the wedge position number and its associated index value. For example, the first wedge displays a zero (0) and will represent the color that is displayed for an index value of zero (0).
4. To change a wedge color, click the wedge, then select a new color by clicking the color button. The Color window opens where you can select a new color. Click OK. The selected wedge is displayed with the new color. You repeat this process for each wedge color that you want to change.
5. To undo any changes that you have made, click the Undo changes button. The color of the selected wedge returns to its previous color.
6. To move a selected wedge forward in the wheel, click the wedge and then click the Move Forward button. The wedge moves forward one slot in the wheel.
7. To move a selected wedge backward in the wheel, click the wedge and then click the Move Back button. The wedge moves backward one slot in the wheel.
8. To add additional wedges to the pie, click the Add button. A new wedge is added to the pie in the last position. It will be added using the color of the currently selected wedge.
9. To remove a wedge, select the wedge that you want to remove from the pie, and click the Remove button.
10. Click Finish. The New Color Sequence wizard closes and the color sequence is displayed in the Color Sequences folder.

Using a color sequence:

To use a color sequence specify the name of a color sequence in a color property cell for an object that you insert in your visual project.

About this task

To use a color sequence:

Procedure

1. Insert an object in your visual project.
2. From the most convenient view, (Properties, Outline or the Project Explorer), click the color property that will use the color sequence.

3. Specify the name of the color sequence that you want to use and the index value. There are several ways that you can specify the index value:
   - Literally: You can use a specific number to indicate the index value. For example:
     \[ \text{colorSeq(Colorseq1, 4)} \]
     The color of this object will be the same color as the wedge labeled index 4 in the color wheel of the color sequence named Colorseq1.
   - With an expression: You can create an expression that can be used to calculate the index value. For example:
     \[ \text{colorSeq(Colorseq1, value1 + 5)} \]
     The color of this object will be from the color sequence named Colorseq1. The index value will be calculated by adding 5 to the value of value1.
   - Using a function: You can use a function to generate an index value. For example:
     \[ \text{colorSeq(Colorseq1, PointNumber())} \]
     The PointNumber() function will be used with a color sequence named Colorseq1. The PointNumber() function assigns a unique color to the object for each row returned by a query.

**Working with color maps**

A color map is a type of global resource that can be used to vary the color of a visual project object depending on a value associated with the object.

**About this task**

For example, you might want a bar chart to display bar colors depending on the values returned by the query. If you want the color to change from red to black when your product shows a profit, you can create a color map that will specify the colors to use when the data reaches certain breakpoint values. Color maps are either discrete or blended. Discrete color maps display a different color for each data value. Blended color maps blend the colors. Breakpoints specify the value at which the color changes.

**Creating a color map:**

You use the New Color Map wizard to create a color map for your visual project.

**About this task**

To create a color map:

**Procedure**

1. You open the New Color Map wizard in one of the following ways:
   - From the Project Explorer view, click the Globals node. Right-click the Color Maps node. Select Insert Color Map from the pop-up menu. The New Color Map wizard opens.
   - From the Project Explorer view, click the Globals node. Double-click the Color Maps node. The New Color Map wizard opens.
2. Type a unique name for the color map in the Name field.
3. Click Distribute. The Color Map Breakpoints window opens.
4. Specify the total number of breakpoints in the Breakpoints list.
5. Specify the minimum breakpoint data value in the Min value field.
6. Type the maximum breakpoint data value in the Max value field.
7. Select the colors you want from the Colors list box. You can choose to set the colors to black, to the colors of the rainbow, or to shades of gray.
8. Click OK. The Color Map Breakpoints window closes and the number of breakpoints that you specified are displayed.
9. Specify the properties of each breakpoint.
   a. Select a breakpoint from the list. The breakpoint value and color are displayed.
   b. Specify the breakpoint value in the Breakpoint field.
   c. Click the Active color box. The Color window opens.
   d. Select the color that you want to display when the breakpoint is reached and click OK. The Color window closes and the new color is displayed in the Active color box.
   e. Click Set. The new value and color are set.
   f. Repeat this process for each breakpoint in the list.
10. For all color maps, click First color to select the first color of the color map. The first color is the color used for all data values that are less than your first breakpoint value.
11. Specify the type of color transition that you want using the Color Transitions radio buttons. Your choices are:
   • Select Discrete to display a pure color from one breakpoint data value to the next breakpoint data value.
   • Select Blend to blend the colors from one breakpoint data value color to the next breakpoint data value color.
12. If you have selected to blend the colors from one breakpoint data value to the next breakpoint data value, then you can select a last color for your color map. The last color is the color used for all data values that are greater than your last breakpoint. The last color and the first color appear un-blended in the color map. To select the last color of a blended color map, click Last color.
13. Click Finish. The New Color Map wizard closes and the color map is displayed in the Color Maps folder.

Using a color map:

To use a color map specify the name of a color map in a color property cell for an object that you insert in your visual project.

About this task

To use a color map:

Procedure

1. Insert an object in your visual project.
2. From the most convenient view (Properties, Outline or the Project Explorer), click the color property that will use the color map.
3. Specify the name of the color map that you want to use and a data value. There are several ways that you can specify the data value:
   • Literally: You can use a specific number to indicate the data value. For example:
colorMap(Colormap1, 10)

The data value 10 will be compared against the breakpoints that have been specified for the color map named Colormap1. The color of this object will be determined by the color assigned to a data value of 10.

- With an expression: You can create an expression that can be used to calculate the data value. For example:

  colorMap(Colormap1,(close_price - open_price))

  The expression will be evaluated based on the current data values. The calculated data value will be compared against the breakpoints that have been specified for the color map named Colormap1. The color of this object will be determined by the color assigned to the calculated data value.

- Using a function: You can use a function to generate a data value. For example:

  colorMap(Colormap1,PointNumber())

  The PointNumber() function will be used with a color map named Colormap1. The PointNumber() function assigns a unique color to the object for each row returned by a query.

Creating a security list

Security lists can be used to tailor visual report or dashboard content to the security level of the user who is viewing the report or dashboard.

About this task

For example, members of the sales team could see sales data, whereas members of the executive team could see sales and profitability data on a particular chart in the dashboard. You can only implement the security list function for visual reports and dashboards that are saved in secured repositories.

To create a security list and implement the security list function:

Procedure

1. You can open the New Security List Wizard in one of the following ways:
2. Type a unique name for the security list in the Name field.
3. Select the users and groups that will be a part of this security list.
4. Click Finish. The security list is added to the Security Lists folder in the Project Explorer view. You can create multiple security lists. Each list can contain a similar or unique set of users and groups. Each list must have a unique name.
5. Insert an object into your visual report or dashboard.
6. Click in the cell of the property that you will control based on the security level of your user. You can choose any property. In order to control what data is displayed to a user, you would select the visibility property for the object.
7. Click the Edit with Expression Designer toolbar button in the Properties view. The Expression Designer opens. Clear the designer work area. Expand the Security node of the function tree. Double-click the function
isSecurityLevel. The template for the function appears as follows in the
designer work area: `isSecurityLevel(strText)`.

8. You will replace the text parameter with a security list. Highlight `strText`
Expand the Security List folder from the designer tree. Double-click the
security list that you want to view the object. The property value appears in
the designer work area as follows: `isSecurityLevel("SecurityList1")`.

9. Click **Apply**.

10. Click **OK**. Only the users that are included in the security list that you
selected can view the object.

**Storing images with visual projects**

A stock image is a type of global resource that enables you to store your image
resources with your visual project.

**About this task**

Storing images with your visual project increases the portability and performance
of the project. Portability of the project is improved because you can run the
project on any machine and still have access to your image as it comes along with
the project. Performance improves because images do not have to be loaded or
reloaded each time they are used.

For example, the processing overhead of loading and reloading a corporate logo
that is used on every page of a visual report or in every scene of a visual
dashboard could be significant. Using a stock image for the corporate logo cuts the
processing time of loading and reloading the image. Pages of a visual report and
scenes from a visual dashboard are displayed to your users more quickly. In
addition, a stock image of the corporate logo eliminates the need for you to hard
code the image location in your visual project.

The following image file formats can be stored as stock images:

- Bitmap (.bmp)
- Device-independent bitmap (.dib)
- Graphic Interchange format (.gif)
- Joint Photographic Experts Group (.jpeg)
- Portable Network Graphics (.png)

**Creating a stock image:**

You can create a stock image for each image that you want to use in your visual
project.

**About this task**

To create a new stock image:

**Procedure**

1. You can open the Open window in one of the following ways:
   - From the **Project Explorer** view, click the **Globals** node. Right-click the
     **Images** node. Select Insert Image from the pop-up menu. An operating
     system specific Open window opens.
   - From the **Project Explorer** view, click the **Globals** node. Double-click the
     **Images** node. An operating system specific Open window opens.
2. From the Open window search for and select the image that you want to use in your visual project.

3. Click Open. A stock image global resource is created. The stock image name is the same as the image name. The new stock image is listed in the Images node in the Project Explorer view.

Using a stock image:

You must associate the stock image in the Images folder with a Picture object in your visual project.

About this task

You can associate a stock image to a Picture object in one of the following ways:

- From the Project Explorer view, click a stock image that you have added to the Images folder and drag the stock image to your visual project canvas. A picture object is created and the ImageSource property is displayed in the Properties view and is automatically updated with the name of the stock image.
- From the Palette view, select a picture object and insert the picture object in your visual project. In the Properties view, click the ImageSource property cell. Type the name of the stock image using the StockImage() function as follows: =StockImage("My_Image") where My_Image is the name of a stock image that you have added to the Images folder for the project.

The picture in your visual project displays the specified stock image. You can then modify additional object properties such as image scaling.

Specifying variable values using parameters

Parameters are a way of specifying variable values for use by your visual project.

About this task

Parameters enable you to do the following tasks:

- Capture user input.
- Capture current context, such as query results, object property values, or locations.
- Pass user input or context information from one part of a dashboard to another.
- Vary query results based on dynamic criteria.
- Vary navigation results based on dynamic criteria.
- Vary visual project design elements based on dynamic criteria.

The following types of parameters are available:

Global Parameters

A parameter that is available to your entire visual project and to the user of your visual dashboard. You can pass global parameters when you specify object properties or when you create event actions. You also have the option of making global parameters public at runtime so they are accessible to the user.

Scene Parameters

A parameter that is available only to a specific scene in a visual dashboard. You can pass scene parameters when you create embedded scenes and event actions.
Query Parameters

Query parameters contain the value that will be sent to the query and used at runtime.

You can use parameters in many ways. Event actions typically capture user input in a parameter. You can use global and scene parameters when you design interactive controls in your scenes; when a user makes a selection in the control, an underlying event action sets the parameter value. The parameter value determines what data is displayed or how it is displayed.

Global, scene, and query parameters contain the following attributes:

- **Name** (parameter names must be unique)
- **Data type**
- **Description** (optional)
- **Default value** (optional)

**Note:** You should provide a default value for your parameters so you can test them.

Working with global parameters:

A global parameter is a type of global resource that is available to your entire visual project. Global parameters enable you to use variable values within your visual dashboard.

Global parameters can either be public or private. A private global parameter is set exclusively within the visual dashboard. A public parameter can be modified by the user at runtime.

DataQuant for Workstation provides the following global parameters:

- **DeviceIdentifier** is a global parameter, which stores the UUID for DataQuant for WebSphere or MAC address for DataQuant for Workstation. When the LoadComplete event occurs, the value of this parameter is defined automatically.

  **Note:** In DataQuant for WebSphere, the UUID is saved to cookies, but if cookies are disabled, the application gets a new UUID every time when the LoadComplete event occurs.

- **ExternalEventID** is a global parameter, set to a particular control on a scene. It saves EventID from **Send to Embedded Scene** event action.

- **Geolocation** is a global parameter that stores the device geolocation data. To get the data, run the [Refresh geolocation event action](#).

- **Translation Table** defines the translation table that is used to display the text values in the visual dashboard.

- **ViewerHeight** contains the height of the current visible part of the scene. This parameter is read-only and its value depends on the default measurement units. This global parameter has been defined for developing visual dashboards. You can use it in expressions when it is convenient to take as a parameter the current height of the visible part of the scene. This parameter value and the expressions with it are automatically updated as you change the size of the current visible scene.

- **ViewerWidth** contains the width of the current visible part of the scene. This parameter is read-only and its value depends on the default measurement units. This global parameter has been defined for developing visual dashboards.
can use it in expressions when it is convenient to take as a parameter the current width of the visible part of the scene. This parameter value and the expressions with it are automatically updated as you change the size of the current visible scene.

- **ViewerX** defines the viewer’s current x-coordinate location, in inches. The default value is zero (0) which translates to the center of the screen. This parameter value is automatically updated as a user repositions in the visual project. This global has been defined for use by visual dashboards.

- **ViewerY** defines the viewer's current y-coordinate location, in inches. The default value is zero (0) which translates to the center of the screen. This parameter value is automatically updated as a user repositions in the visual project. This global has been defined for use by visual dashboards.

- **ViewerZoom** defines the viewer’s current zoom level. The default value is 100%. This parameter value is automatically updated as the user changes zoom levels.

You can define your own global parameters.

*Adding a global parameter:*

You can define your own global parameters.

*About this task*

To add a user-defined global parameter:

*Procedure*

1. You can open the Insert Parameter window in one of the following ways:
   - From the **Project Explorer** view, expand the **Globals** node. Right-click the **Parameters** node. Select **Insert Parameter** from the pop-up menu.
   - From the **Project Explorer** view, expand the **Globals** node. Double-click the **Parameters** node. The Insert Parameter window opens.
2. Type a unique name for the global parameter in the **Name** field.
3. Select a data type from the **Data type** list box. The data type will describe what type of data will be included in the parameter. Your choices are:
   - **Boolean**: True or False
   - **Color**: Color
   - **DateTime**: Date and time value
   - **DateTimeSet**: Multiple data and time values
   - **FilePath**: Path to a file
   - **FontName**: Valid font type
   - **Integer**: Whole number
   - **Literal**: As is data. This data type only applies to query parameters.
   - **Number**: Floating point number
   - **NumberSet**: Multiple floating point numbers
   - **Percentage**: Value that will be used as a percentage
   - **Point**: Single x,y point value
   - **PointSet**: Multiple x,y point values
   - **PolySet**: Multiple point variables that can be used to construct a polygon
   - **Text**: A text string
   - **TextSet**: Multiple text strings
4. To add a description, type the text in the Description field.
5. To specify that the parameter be available to the user, select the Public at runtime check box. When checked, a Runtime Settings window will be issued at runtime and the user can enter information.
6. To specify a default value, select the Has default value check box. A field is displayed where you can type a default value for the global parameter.

Using global parameters:

Global parameters can be used to specify object properties in visual projects.

Before you begin

About this task

To use global parameters when specifying object properties:

Procedure

1. Insert an object in your visual project.
2. From the Properties view, select the property that will derive its value using your global parameter and type =your_global_parameter_name in the corresponding property cell.
3. You can also use a global parameter in an expression that will be evaluated in order to derive the property value. Open the Expression Designer. The global parameters that have been defined are listed in the left pane.

Working with Expiration Schedules

Use expiration schedules to specify when the data that is related to your visual project must expire.

About this task

Expiration schedules can be set up to automatically refresh the data that is called in a visual project. If an expiration schedule is assigned to a query in a visual project, the query results that are obtained from the query are saved for the given amount of time that the schedule dictates. When a user runs the visual report or dashboard, the query results that are saved in the cache are used and the query is not rerun. Once the time and date set up by the schedule is reached, the query results expire, and the next time a user runs the visual report or dashboard, the query is rerun. Expiration schedules can save time and database resources by only running queries when updated data is needed.

Note: You can use expiration schedules only for the visual projects that are saved to repository.

To create a new expiration schedule:

Procedure

1. The Expiration Schedule Configuration wizard can be opened in one of the following ways:
   • From the Project Explorer view, expand the Globals node. Right-click the Expiration Schedules node. Select Insert Schedule from the pop-up menu. The Expiration Schedule Configuration wizard opens.
• From the Project Explorer view, expand the Globals node. Double-click the Expiration Schedules node. The Expiration Schedule Configuration wizard opens.

2. Enter a name for the expiration schedule in the Schedule name field.

3. Select how often the schedule expires in the Data expires radio group. Valid choices are:
   • hourly to have the schedule expire every hour.
   • daily to have the schedule expire every day.
   • weekly to have the schedule expire every week.
   • monthly to have the schedule expire every month.
   • at specific date to have the schedule expire only at a specific date.
   • never expires to never have the schedule expire. The same query results will be used every time the project is run.
   • always expired to always have the schedule expired. The query results will never be saved and the query will be rerun every time the project is run.
   • after the visual project is closed to have the schedule expire after you close the visual project.

4. If you selected never expires or always expired, click Finish. The Expiration Schedule Configuration wizard closes and the new expiration schedule is saved under the Expiration Schedules node in the Project Explorer view.

5. If you selected any of the other options, click Next. The Set schedule properties page of the Expiration Schedule Configuration wizard is different depending on your selection.

6. Specify the Set schedule properties options and click Finish. The Expiration Schedule Configuration wizard closes and the new expiration schedule is saved under the Expiration Schedules node in the Project Explorer view.

Specifying hourly expiration schedule options:

Hourly expiration schedules expire every hour.

About this task

To set the options for an hourly expiration schedule:

Procedure

1. Open the Set schedule properties page of the Expiration Schedule Configuration wizard.

2. Specify the date that the schedule will begin to expire in the Begin date field.

3. Specify at which time on the specified date the schedule will begin to expire in the Begin time field.

4. Set the expiration interval by specifying the number of hours and minutes in the Expiration interval group of fields.

5. Click Finish. The Expiration schedule configuration wizard closes and the new expiration schedule is saved under the Expiration Schedules node in the Project Explorer view.

Specifying daily expiration schedule options:

Daily expiration schedules expire on a day to day basis.
About this task

To set the options for a daily expiration schedule:

Procedure
1. Open the Set schedule properties page of the Expiration schedule configuration wizard.
2. Specify the date that the schedule will begin to expire in the Begin date list box.
3. Specify at which time on the specified date that the schedule will begin to expire in the Begin time list box.
4. Specify an option in the Data expires radio group. Valid options are:
   - daily: the schedule will expire every day at the specified begin time, starting at the specified begin date.
   - each: the schedule will expire after the number of days specified in the day list box, at the specified begin time, starting at the specified begin date.
   - days of week: the schedule will expire every day of the week that is checked, at the specified begin time, starting at the specified begin date.
5. Click Finish. The Expiration schedule configuration wizard closes and the new expiration schedule is saved under the Expiration Schedules node in the Project Explorer view.

Specifying weekly expiration schedule options:

Weekly expiration schedules expire on a week to week basis.

About this task

To set the options for a weekly expiration schedule:

Procedure
1. Open the Set schedule properties page of the Expiration schedule configuration wizard.
2. Specify the date that the schedule will begin to expire in the Begin date list box.
3. Specify at which time on the specified date that the schedule will begin to expire in the Begin time list box.
4. Specify how many weeks from the specified begin date until the schedule expires in the each week list box.
5. Specify on which days of the specified week that the schedule will expire in the days of week check box group.
6. Click Finish. The Expiration schedule configuration wizard closes and the new expiration schedule is saved under the Expiration Schedules node in the Project Explorer view.

Specifying monthly expiration schedule options:

Monthly expiration schedules expire on a month to month basis.

About this task

To set the options for a monthly expiration schedule:
Procedure
1. Open the Set schedule properties page of the Expiration schedule configuration wizard.
2. Select the date that the schedule will begin to expire in the **Begin date** list box.
3. Select at which time on the specified date that the schedule will begin to expire in the **Begin time** list box.
4. Specify an option in the **Data expires** radio group. Valid options are:
   - **Day** to have the schedule expire on the day of the month specified in the **day** list box.
   - **or** to have the schedule expire at the same relative day of the month. In the first drop-down list, you specify that the schedule will expire on the first, second, third, fourth, or last day that you specify in the second drop-down list.
5. Specify which months the schedule will expire in the **Select months** check box group.
6. Click **Finish**. The Expiration schedule configuration wizard closes and the new expiration schedule is saved under the **Expiration Schedules** node in the **Project Explorer** view.

Specifying options for schedules that expire on specific dates:
Expiration schedules can be set to expire only on specific dates.

About this task

Procedure
1. Open the Set schedule properties page of the Expiration schedule configuration wizard.
2. Click the first row of the **Date and time** list. Click the button with the three dots on the right. The Select date and time window opens.
3. Use the left and right arrow buttons to select the month and year that the schedule will expire.
4. On the calendar, select the day of the month that the schedule will expire.
5. In the **Time** field, select the time at which the schedule will expire.
6. Click **OK**. The Select date and time window closes and the new date and time is stored in the **Date and time** list.
7. You can repeat these steps to specify multiple dates and times when the schedule will expire.
8. Click **Finish**. The Expiration schedule configuration wizard closes and the new expiration schedule is saved under the **Expiration Schedules** node in the **Project Explorer** view.

Working with translation tables
Translation tables enable visual designers to publish a single visual dashboard that can be viewed in multiple languages.

About this task
When content developers create a dashboard, button captions, labels, tooltips and other embedded text is entered directly into the property values of the dashboard objects. Using translation tables, content developers can supply alternate versions of this entered text, to be displayed when the user is in a different locale. For
example, a Japanese developer can elect to author a new dashboard in Japanese but then also include translation tables for German and French, each of which will be used when the dashboard is viewed under German and French locales.

Every new dashboard is created with a default translation table. The default translation table provides the collection of supplied text that has been added directly within the property values of the components that make up the dashboard. The contents of the default translation table will be used, unless there is an alternate text value available in a translation table that is more appropriate for the current locale. Consider the following example:

1. A dashboard is authored in Japanese and includes seven text strings.
2. The dashboard developer decides to include a German translation table, providing German translations for three of the seven original text strings. The developer also adds a French translation table, providing translations for all seven strings.
3. When the dashboard is viewed in Japan, all seven original text values are displayed. When viewed in Germany, the three translated strings are used, along with the four original Japanese strings. When viewed in France, the seven French translated strings are used. When viewed in Italy, the Japanese strings are used.

Any number of translation tables can be added to a dashboard project, each of which is associated with a specific language. Although it is possible to directly edit the strings within the translation table editor, the application also provides a means of exporting and re-importing translation tables, allowing content developers to perform the translation using external tools and applications.

**Procedure**

1. From the **Project Explorer** view, expand the **Globals** node for the dashboard. Right-click the **Translation Tables** node. Select **Insert New Table** from the pop-up menu. The New Translation Table wizard opens.
2. Select a language from the **Language** drop-down list. The language that you select will be used as the new translation table name. Only one translation table can be defined for a given language. If a translation table already exists for a selected language, an error message will appear over the pull-down menu.
3. Click **Finish**. The New Translation Table wizard closes, and the Edit Translation Table window opens.
4. The new translation table is populated with the contents of the default translation table. An object's text property value ID is listed in the **ID** cell of the translation table. The text that has been defined for the object's text property value is listed in the **Text** cell of the translation table.
5. Using the check boxes on the left, clear the check box that is associated with each ID that will be excluded from translation. Only the values for the IDs that remain checked will be translated. Dashboard designers will recognize which values should be translated. Some text values such as company names do not need translating so they should be excluded. Some text property values are not simply strings but calculated (displayed in the text cell with an = sign). If these objects include string values, they should be included for translation. When an object ID is disabled in a translation table, the corresponding value in the default translation table will be used (text values will be in the original language used to create the dashboard). You can use the **Enable** icon to enable one or more objects if they have been disabled.
6. You can choose to translate the strings within the Edit Translation Table window. You can also export the strings for translation to a file. To export the strings to a file, select Export. An operating system specific Save window opens.

7. Browse to the directory where you will save the file and click Save. The file name can be different from the language under which the translated table will be saved. Make the file accessible to the translators.

8. The following is an example of how the strings appear in the exported file: TestDashboard.Level1.Text10.Text=Current Assets. Translators will translate only the actual text in the string and not the syntax of the object ID. This text appears to the right of the equal sign (=). In the example, only Current Assets should be translated.

9. When the translation is complete, you will upload the file. From the Project Explorer, right-click the translation table that you will be uploading, select Edit Table from the pop-up menu. Click Import. An operating system specific Open window opens.

10. Navigate to the translated file and click Open. The new changes are shown in the Text column of the Edit Translation Table window.

11. Click OK. The new translation table is saved and its corresponding language is listed under the Translation Tables node for the visual dashboard.

12. In the Properties view of the active visual dashboard, specify the Default language of the dashboard. Valid options include:
   - Auto: The native language of the machine on which DataQuant for Workstation was installed.
   - Default: The original language used to create the visual dashboard content.
   - If any translation tables have been created for the selected visual dashboard, their corresponding languages will also be listed as default language options.

Switching translation tables in the scene of a dashboard:

You use the Translation Table global parameter to allow users to change the translation of text values in a dashboard from a scene.

About this task

There might be a time when a dashboard user from a certain locale wants to select a different language in which to view text than the default language. You can use the Translation Table global variable to create controls that allow a user to switch from one translation table to another. The following steps describe one common way in which to create a user interface that gives a user this ability:

Procedure

1. Create translation tables for each language that you want to include as an option in your dashboard.

2. From the Palette view, select the list box object from the Controls palette and click and drag the shape in the editor window. The ListBox Wizard opens.

3. Select No and click Next. The Enter option and value list page opens.

4. Click Finish. The ListBox Wizard closes and the list box displays in the editor window.

5. In the Properties view, select the OptionList property and click the Edit with Expression Designer button in the toolbar. The Expression Designer opens.
6. Expand the Translation folder in the functions tree and double-click the getAvailableTranslationTables() function.
7. Click OK. The Expression Designer closes and all of the translation table languages that you created are listed in the list box.
8. From the Palette view, select the button object from the Controls palette and click and drag the shape in the editor window.
9. With the button selected in the editor, open the Events view and double-click the Click event. The [Button Name] Behavior window opens.
11. Double-click the Set values action. The Set values action parameters page opens.
12. Click the Add New Entry button and then click the ellipsis (...) in the Parameter cell. The Expression Designer opens.
13. From the Parameters folder of the Globals node, double-click Translation Table.
14. From the ListBox1 properties folder of the ListBox1 node, double-click the Selection property and click OK. The Expression Designer closes and control returns to the Add New Action window.
15. Click Finish. The Set values action is added. The Add New Action window closes.
16. From the [objectname] Behavior window, click the Apply button to apply all actions that have been added for the event.
17. Click OK to close the [objectname] Behavior window.
18. Click the Runtime tab, select a language from the list box, and click the button. The next time a process is run in the dashboard (such as running a query) the text values of the scene should display in the selected language.

---

**Profiling visual dashboards**

You use profiling to analyze the complexity of visual dashboards scenes and objects they contain dynamically; to measure the data on query running and objects creating and also the time spent on compiling the whole visual project.

**About this task**

With the help of profiling you can monitor the work of scenes and objects and find the problematic ones. Using this information you can optimize your visual dashboards.

**Analyzing the complexity of a scene**

You can analyze the complexity of a particular dashboard scene with the help of the Analyzer view.

**Before you begin**

You must have a visual project to analyze and open the Analyzer view.

**About this task**

Analyzer counts the weight of each object lying on a scene. Each object has a number of properties that contain values and expression of different complexity. All default properties are basic units and cannot be subdivided into simpler ones.
Each default property has the weight that equals 1. The weight of non-default properties is counted as an expression according to their complexity. In the analysis process you can count the weight of each object and get the overall weight of a scene.

The objects that can run queries or the DataTemplates have incomputable weight. They are highlighted in red in the Analyzer view.

The analysis data is represented in a table with the following columns:
- **Elements** column displays a tree structure of the analyzed scene.
- **Weight** column displays a weight of a particular element.
- **Cumulative weight** column displays a weight of a particular parent and the child elements.

To analyze a scene:

**Procedure**
1. Open the scene you want to analyze in Canvas, Design, or Runtime mode.
2. In the Analyzer view click Analyze.
3. The analysis of the scene is displayed in the Analyzer view as a tree listing all the components and objects of the scene.
4. You can select an object in the view and it will be displayed as selected on the scene and in the Project Explorer view.
5. You can specify the values for objects with incomputable weight by clicking the ellipsis (...) in the Cumulative Weight column or Set row count button on the view toolbar. This value will be used as an average row amount in the incomputable object.

**Profiling a visual dashboard**

You can profile a visual dashboard with the help of the Profiler view.

**Before you begin**

You must have a visual project to analyze and open the Profiler view.

**About this task**

The Profiler view displays the data on query running, the amount of created objects, and the time spent on creating them and calculating expressions on the visual project. The profiler data is represented in a table with the following columns:
- **Elements** column lists all the scenes with the objects, expressions and queries of a visual project.
- **Count** column shows the number of times the object was created.
- **Time** column displays the time spent on creating an element.
- **Cumulative time** column displays the time spent on creating a parent and its child elements.
- **Percentage** column displays the percentage ratio of the time spent on creating a particular element to the time spent on creating the entire project.

To start the profiling process:
Procedure

1. Click the Profile ( ) button on the Profiler view toolbar.
2. Open the required visual dashboard or a separate scene in the Runtime mode.
3. When all scenes are created and all queries are run click the Profile ( ) once again. The profiler results are displayed in the tree table.
4. You can modify the way the results are displayed by clicking Tree view ( ) or Flat view ( ) buttons on the Profiler view toolbar.
5. To view the SQL text of a query in the project, select the query node in the Profiler view and click the SQL Text ( ) toolbar button.
6. You can export or import the profiler data by clicking the Import data ( ) or Export data ( ) toolbar buttons.
7. You can select an object in the view and it will be displayed as selected in the Project Explorer view.
8. You can sort the values in the Profiler view columns by clicking their headers.

Debugging visual dashboards

You can control the execution of the processes and check for the errors that can exist in visual dashboards using the Expressions view. You can check parameter values, object properties, and expression evaluations for the appropriate scene of the visual project in Runtime mode.

About this task

To check a scene, do the following steps:

Procedure

1. Open the Expressions view. To do this, from the main menu, select Window > Show View > Other, expand the Visual Projects Debugger node, select Expressions, and click OK.
2. From the Expressions view, click Watch Global Parameters to check the values of the global parameters of the visual project for the scene when you run it.
3. Click Watch Local Parameters to check local parameters of the scene when you run it.

   Note: Global and local parameters specific only for the scene that you run are displayed.
4. You can add an expression that can contain functions, refer to the object properties or scene parameters and watch the evaluation of this expression. Click Add Expression to open the Add Expression dialog.
5. Type the text of the expression and click OK to add this expression to the Expression list.

   Note: The added expressions are evaluated for all the scenes that you run.
6. To change the existing expression, select it from the Expression list and click Edit Expression. The Edit Expression dialog with the text of the selected expression opens.
7. Edit the text of the expression and click OK.
Tip: If you have a scene open in either Canvas, Design, or Runtime mode, the Expression Designer window opens by clicking Add Expression or Edit Expression. You can enter expressions using the expression designer functions.

8. To watch the values of the scene object properties, right-click the object in either Canvas mode, Design mode, or Project Explorer view, and select Watch Properties from the menu. The Select Properties to Watch dialog opens.

9. Select check boxes corresponding to the object properties that you want to watch and click OK. The properties are added to the Expression list.

10. Open the scene that you want to check in Runtime mode. In the Value list of the Expressions view, check the global and local parameters and evaluations of the expressions for the current scene.

11. Click Refresh Expressions to update the values of the expressions for the active scene.

12. To delete the expression from the Expression list and stop watching it, select it from the list and click Remove Expression.

13. Click Clear Expressions to delete all expressions that are watched from the Expression list.

---

**Editing queries of particular objects in visual projects**

You can quickly view and edit a query that is used by a particular object in the visual project.

**Before you begin**

The query used by the object of the visual project must have parameters.

**About this task**

To open and edit a query of a particular object in a visual project, perform the following steps:

**Procedure**

1. From the Project Explorer view, expand the node of the object for which you want to view the specified query.

2. In the tree of the object, expand the QueryParameters node and select the item containing query parameters. The names of the parameters and their current values are displayed in the Properties view. You can change the values of the query parameters.

3. Right-click the selected item with query parameters and select Open Query from the menu. The query used by the object is opened in the Query editor. You can modify the query in this editor.
Chapter 14. Working with JavaScript Tables

You can use JavaScript Tables to collect the data from network resources and process it in the form of relational tables.

For example, you need to collect the stock exchange information from a financial portal for a certain period of time and save this information so that you can use it in DataQuant as a typical relational table.

For this, you create a JavaScript Table, specify its structure, and save it to a virtual data source. In the JavaScript Table editor, you write a script that will populate the table with the required data retrieved over HTTP or HTTPS. When you run a query based on the created JavaScript Table, the script retrieves the data and saves it in a table that is used by the query.

And if it is necessary, you can use external functions and objects from different modules either saved in a repository or located on a web server.

You can run queries based on a JavaScript Table, use them to create quick reports, forecasts, or add to visual dashboards.

Creating JavaScript Tables

You use the Create JavaScript Table wizard to add a new JavaScript Table and specify its structure.

**About this task**

To create a JavaScript Table:

**Procedure**

1. Select File > New > Other. In the opened window select JavaScript Tables and click Next. The Create New JavaScript Table wizard opens.

2. On the Add New JavaScript Table page, specify the name for the table you create and select a virtual data source where to save it. Click Next.

   **Note:** The Data Source list contains only virtual data sources from the current repository connection.

3. On the JavaScript Table Structure page, define the table structure including the list of columns, their order, value types and required parameters. You can also add comments to any column.

4. Click Finish to close the wizard. The JavaScript Table with the specified name and structure is added to the selected virtual data source. After closing the wizard, JavaScript Table editor opens.

5. In the JavaScript Table editor, you create scripts to populate the JavaScript Table with the required data.

6. When you finish working with script, save the changes and close the editor.
Results

Now you can use the created JavaScript Table as a typical relational table. For example, you can run queries based on the created JavaScript Table, use them to create quick reports, forecasts, or add to visual dashboards.

Writing scripts to populate JavaScript Table

Once you created a JavaScript Table and specified its structure, you need to write a script that will populate the table with the required data.

About this task

You write the script in the JavaScript Table editor according to general JavaScript syntax rules. The editor opens automatically after you close the Create New JavaScript Table wizard.

To manage the content of a JavaScript Table, you can also use the following special functions of the table global object:

- replaceData()
- appendData()
- eraseData()
- setTag()
- getTag()
- include()

To trace HTTP request and response and record logs, use log() function.

You can also use prompted variables in the script. A user will be prompted to enter variable values when running a query based on the created JavaScript Table. If you use a prompted variable in the script, note the following peculiarities:

- The JavaScript Table cache is ignored.
- When a user works with a JavaScript Table in DataQuant for WebSphere, all the retrieved data will be available to that user only.

When you finish working with the script, save the changes and close the editor.

replaceData()

Completely replaces the content of a JavaScript Table with the data specified by a two-dimensional array of the following form: `[[<row 1 column 1>, <row 1 column 2>,...], [<row 2 column 1>, <row 2 column 2>,...],...]`.

Note: The structure of input array must correspond to the structure of the JavaScript Table.

Syntax

replaceData(<two-dimensional array>);

Example

```javascript
var data = [['Property_1', 50], ['Property_2', 60]];
table.replaceData(data);
```

appendData()

Inserts the data specified by a two-dimensional array to the end of the JavaScript Table.

```javascript
table.appendData(data);
```
AppendData

appendData(<two-dimensional array>);

Example

```javascript
var data = ["Property_1",50], ["Property_2",60];
table.appendData(data);
```

EraseData

Removes JavaScript Table records that satisfy the specified SQL condition. If you call eraseData() function without arguments or with null argument, all table records are removed.

Syntax

```javascript
eraseData(<SQL condition>);
```

Examples

Example 1

```javascript
table.eraseData("ID < 100 AND DEPT = 15 OR SALARY IS NULL");
```

Example 2

```javascript
function eraseOutdated()
{
  var expirationDate = new Date();
  expirationDate.setDate(-30);
  var day = expirationDate.getDay();
  var month = expirationDate.getMonth() + 1;
  var year = expirationDate.getYear() + 1900;
  var strDate = year + '-' + month + '-' + day;
  table.eraseData("Last Trade Date"<" + "'" + strDate + "'";)
}
```

SetTag

Links the JavaScript Table with some additional information specified by a string. This information can be retrieved by the getTag() method. To clear the information, use setTag() with null argument or without arguments.

Syntax

```javascript
setTag(<string>);
```

Example

```javascript
function setTableModificationTime()
{
  var currentDate = new Date();
  table.setTag(currentDate.getTime());
}
```

GetTag

Returns the additional information related to the JavaScript Table.

Syntax

```javascript
getTag();
```
Example

```javascript
function getTableModificationTime() {
    var tag = parseInt(table.getTag());
    return tag;
}
```

**include()**

You can include external code in your JavaScript Table script by using the include() method.

**Note:** You can use external functions and objects only after connecting the required external module to your script with the include() method.

**Syntax**

```javascript
include(strLocation);
```

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>strLocation</td>
<td>Specifies either a path to a JavaScript Module in a repository or a link to external source from the Internet.</td>
</tr>
</tbody>
</table>

**Examples**

Example 1. External module from the Internet containing the `algorithm()` method:
```javascript
table.include('http://example.com/algorithm.js');
table.replaceData(algorithm());
```

Example 2. JavaScript Module from the `rsbi:/workspaces/Default/algorithm` repository contains the `algorithm` method:
```javascript
table.include('rsbi:/workspaces/Default/algorithm');
table.replaceData(algorithm());
```

**log()**

Displays messages in the Output view and writes them to DataQuant log.

**Note:** The `log.metadatalayer.javascript` logger category must be set to `info` or higher.

**Syntax**

```javascript
log();
```

**Examples**

```javascript
log(data);
log("Data is already up to date");
```

**RSBIHttpRequest class**

The class that is used to make HTTP requests. It has the same functions as the standard JavaScript object XMLHttpRequest 1.0. The latter is also supported but it is recommended to use RSBIHttpRequest.

**NTLM authentication**

You can specify the additional parameters in the `open()` method of RSBIHttpRequest class to use NTLM authentication. See Example 2.

**Parameters**
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>method</td>
<td>Specifies the HTTP method to use, such as &quot;GET&quot;, &quot;POST&quot;, &quot;PUT&quot;, &quot;DELETE&quot; and so on.</td>
</tr>
<tr>
<td>url</td>
<td>Specifies the URL to send the request to.</td>
</tr>
<tr>
<td>async</td>
<td>Supplies a Boolean value indicating whether or not to perform the operation asynchronously.</td>
</tr>
<tr>
<td>username</td>
<td>Required for NTLM authentication. This parameter specifies the name of the user.</td>
</tr>
<tr>
<td>password</td>
<td>Required for NTLM authentication. This parameter specifies the password of the user.</td>
</tr>
<tr>
<td>domain</td>
<td>Required for NTLM authentication. This parameter specifies the name of the domain.</td>
</tr>
<tr>
<td>workstation</td>
<td>Optional for NTLM authentication. This parameter specifies the name of the workstation.</td>
</tr>
</tbody>
</table>

**Examples**

Example 1

```javascript
var request = new RSBIHttpRequest();
request.open('GET', 'http://example.com', false);
```

Example 2

```javascript
var req = new RSBIHttpRequest();
req.open("GET", 'http://example.com', false, "username", "password", "domain", "workstation");
```

or

```javascript
req.open("GET", 'http://example.com', false, "username", "password", "domain");
```

**JavaScript Table Example**

The following example demonstrates how to use JavaScript Tables.

**Purpose**

You need to collect stock exchange information from a financial portal for a certain period of time and save this information so that you can use it in DataQuant as a typical relational table. The table must not be updated too often and store information longer than 30 days.

**Preparing table structure**

Before you begin writing the script, you need to make sure that the specified table structure corresponds to the data that you need to retrieve.

**Planning the script structure**

According to the purpose the script must retrieve information from server, parse the data to fill the table, save and check the last modification time, and remove outdated information.

```javascript
var lastModificationDate = getTableModificationTime();
if (needUpdate(lastModificationDate))
{
    var raw_data = retrieveRawData(url);
}
```
var data = parseData(raw_data);
table.appendData(data);
eraseOutdated();
setTableModificationTime();
}
else
{
  log("Data is already up to date ");
}

The following functions are used:
- `setTableModificationTime()` to attach the last modification time to the table.
- `getTableModificationTime()` to get the last table modification time
- `needUpdate()` to check if update is required depending on the last modification time.
- `retrieveRawData()` to get the required data from the portal.
- `parseData()` to split the received string according to the table structure.
- `eraseOutdated()` to remove old records.
- `log()` to trace the response status and write information to the Output view.

Retrieving data from the Internet

First of all, you need to retrieve the required data from the Internet. For this, `RSBIHttpRequest` class is used:

```javascript
function getStockData(url)
{
  var request = new RSBIHttpRequest();
  request.open('GET',url, false);
  var response = '';
  request.onreadystatechange = function()
  {
    if(request.readyState === 4)
    {
      response = request.responseText;
    }
  }
  request.send(null);
  return response;
}
```

The `getStockData()` function returns the response string with all required data from the portal.

It is called from the `retrieveRawData()` function that forms an HTTP request and uses `log()` function to trace response information. You can also use prompted variables (the `symbols` variable) in script – user will be prompted to specify the values when running the query. The `&DSQQW_DQ` global variable is used for substitution of double quotes.

```javascript
function retrieveRawData()
{
  var symbols = &DSQQW_DQ&symbols&DSQQW_DQ;
  var url = "http://finance.yahoo.com/d/quotes.csv?s="+symbols+'&f=snl1d1t1ohgdr';
  var raw_data = getStockData(url);
  log(raw_data);
  return raw_data;
}
```
Parsing received data

Secondly, you need to parse the retrieved data according to your purposes so that you can fill the table with the help of `appendData()` or `replaceData()` functions. The easiest way is to split the response string:

```javascript
function parseData(raw_data) {
    var data = [];
    var rows = raw_data.split('\n');
    for (var i=0; i < rows.length; i++) {
        var row = rows[i].trim();
        if (row !== '') {
            var cells = row.split(','),
                data.push(cells);
        }
    }
    return data;
}
```

Specifying data update interval

You can attach modification time (as any additional information in string format) to the table with the help of `setTag()` function. The `getTag()` function is used to retrieve this information.

1. Setting the current time (last modification) as a table tag:
   ```javascript
   function setTableModificationTime() {
       var currentDate = new Date();
       table.setTag(currentDate.getTime());
   }
   ```

2. Retrieving the last modification time:
   ```javascript
   function getTableModificationTime() {
       var tag = table.getTag();
       if (tag === null) {
           var d = new Date();
           d.setDate(-2); // to force update for the first time use
tag = d.getTime();
       } else {
           tag = parseInt(tag);
       }
       return tag;
   }
   ```

3. Checking whether update is required:
   ```javascript
   function needUpdate(lastModificationDate) {
       var yesterday = new Date();
       yesterday.setDate(-1);
       return lastModificationDate <= yesterday.getTime();
   }
   ```

Removing outdated information

You can use the `eraseData()` function to remove outdated information from the table:
function eraseOutdated()
{
    var expirationDate = new Date();
    expirationDate.setDate(-30);
    var day = expirationDate.getDay();
    var month = expirationDate.getMonth() + 1;
    var year = expirationDate.getFullYear() + 1900;
    var strDate = year + '-' + month + '-' + day;
    table.eraseData("Last Trade Date" + ' ' + strDate + ' ');
}

Displaying results

When you finish working with the script, save the changes and close the editor.

To run the query, navigate to the saved JavaScript Table in the Repository Explorer
view and double-click it. Run the opened query.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Name</th>
<th>Last Trade (Price Only)</th>
<th>Last Trade Date</th>
<th>Last Trade Time</th>
<th>Open</th>
<th>Day's High</th>
<th>Day's Low</th>
<th>Dividend/Share</th>
<th>P/E Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>XOM</td>
<td>96.54/2013-09-02</td>
<td>&lt;null&gt;</td>
<td>96.42</td>
<td>95.92</td>
<td>95.42</td>
<td>96.02</td>
<td>0.28</td>
<td>9.32</td>
</tr>
<tr>
<td>2</td>
<td>GE</td>
<td>23.29/2013-09-02</td>
<td>&lt;null&gt;</td>
<td>23.29</td>
<td>23.57</td>
<td>23.09</td>
<td>23.59</td>
<td>0.72</td>
<td>17.80</td>
</tr>
<tr>
<td>3</td>
<td>MSFT</td>
<td>88.61/2013-09-02</td>
<td>&lt;null&gt;</td>
<td>88.61</td>
<td>88.50</td>
<td>88.35</td>
<td>88.50</td>
<td>0.86</td>
<td>55.72</td>
</tr>
</tbody>
</table>

Editing JavaScript Tables

You can edit the structure and the script of already created JavaScript Table.

About this task

To edit JavaScript table:

Procedure

1. In the Repositories or Repository Explorer view, navigate to the required
JavaScript Table.

2. To edit the structure of the JavaScript Table, right-click this table and select
Properties from the popup menu.
   a. Open the Structure page in the Properties window.
   b. Change the table structure including the list of columns, their order, value
types and required parameters. You can also add comments to any column.
   c. To cancel unsaved changes, click Restore Defaults.
   d. To save the updated structure of the JavaScript Table, click Apply, and then
      OK.

   Note: All the data will be removed from the table.

3. To edit the script of the JavaScript Table, right-click this table and select Edit
Source Script from the popup menu. In the opened JavaScript Table editor,
rewrite the script according to your purposes. Once you finished editing the
source script, save the JavaScript Table.
Chapter 15. Using Text Analytics

You can use the DataQuant Text Analytics feature to analyze and interpret the relevance of words and phrases cited across multiple unstructured texts. This functionality is available for Windows operating systems only.

Text Analytics perspective

The Text Analytics perspective provides the interface from which you create and work with Text Analytics projects.

From the Text Analytics perspective you can add unstructured texts to a project and extract data analytically. The Text Analytics feature applies data mining and extraction techniques against unstructured text documents by using rules-based knowledge engine.

Opening the Text Analytics perspective

To open Text Analytics perspective, select Window > Open perspective > Other > Text Analytics.

Text Analytics perspective views

The Text Analytics perspective consists of the text viewer in the editor area and several views, which are defined below:

- **Highlighter**
  This view displays the text with the highlighted word selected in the Word Map view.
  Use this view to scroll the text with the highlighted word and find its textual context.
- **Sources Explorer**
  This view displays the tree structure of the files, belonging to the projects.
  Use this view to add folders to or delete folders from a project and open word matches in the Highlighter view.
- **Text Analytics Explorer**
  This view displays the lists of Text Analytics and Simple Text Analytics projects that can be run to analyze the specified documents.
  You use this view to manage a project.
- **Entities Explorer**
  If you work with Text Analytics projects, this view displays the tree structure of the entities which are found on the basis of knowledge base rules.
  If you work with Simple Text Analytics projects, this view displays the following tree branches:
  - Predefined dictionaries. This branch lists the built-in dictionaries if you decide to use them for text analysis process.
  - User dictionaries. This branch lists your own dictionaries that you specify in the New Dictionary wizard.
  Use this view to select the necessary nodes and display the appropriate analysis results in the Word Map view.
Customizing the knowledge base

You can customize the knowledge base by setting load values and processing sequences.

About this task

A knowledge base provides a set of rules by which the Text Analytics feature analyzes text. These rules contain lexical information and patterns that form the basis for the analysis.

Through customization you can control some of the knowledge base processing capabilities. For example, you can augment the scope of the knowledge base by adding extensions and you can enhance performance by specifying loads and processing sequences.

DataQuant includes a sample knowledge base, which is in the following directory:

DataQuant for Workstation install_directory\Sample AeroText KB\sample.kb

You can customize the sample knowledge base or any other existing knowledge base to provide a more tailored analysis of unstructured text documents.

Note: This task does not describe how to create or configure the knowledge base itself. If you want to modify the rules in the sample knowledge base, or if you want to create a new knowledge base, you have to use the Rocket AeroText product. For information about Rocket AeroText, go to http://www.rocketsoftware.com/products/aerotext(descriptions).

It is assumed that the person performing this task has an understanding of knowledge bases.

To customize the knowledge base:

Procedure

1. Open the New AeroText Knowledge Base Configuration wizard using one of the following methods:
   • Select File > New > Other > AeroText Knowledge Base Configuration
   • From the Text Analytics perspective, click File > New > AeroText Knowledge Base Configuration

2. In the Configuration name field, enter the name of the customized configuration and click Finish. By default, the Main view opens.

3. Specify the knowledge base to configure in the Knowledge Base field by clicking the ellipsis (...) button or by typing the path to the knowledge base file.
   All the tab controls are disabled until you specify the Knowledge Base field. The Tokenizer is a read-only field. It displays a name of the file with a tokenizer (for example, *.dll library for Windows).
4. Specify a predefined rule from the knowledge base in the **Rule slot** field.
   - The set of rules in the knowledge base includes actions that fill rule slots with extracted information.
   - Rule slots are used to hold extracted text and sometimes related information.
   - If this slot is found in the knowledge base, this rule will be included in the results and be displayed in the Entities Explorer tree.

5. Specify the content source in the **Content from** area.
   By default, content is taken from the Summary. If you want the source to come from a specific slot, select Content slot and enter the slot name.

6. Use the icons in the **Knowledge Base loads** pane to set load information.
   You can name a particular load in the Name field, add a comment in the Comment field, select the correct rule base and post action in the Rulebase, and Post action lists.

7. Use the icons in the **Knowledge Base Processing sequence** pane to set the processing sequence.
   You can name a particular sequence of actions in the Name field, add a comment in the Comment field, select the actions from the All actions list and add them to the Active actions list. The selected actions are applied to analyze the documents.
   Each action in the processing sequence defines which cache or parts of a cache will be given to the pattern matcher and which rule base will be run against that cache.
   A cache is a container that can store one or more records. If any caches are to be attached (i.e., pattern matching run against the contents of the cache), they are specified in the action at this level. Information obtained during rule matching is stored in a cache by the rule actions.

8. Click the **Extensions** tab to view the Extensions editor area.
   You can use this editor to extend dictionaries in the knowledge base.

9. Select the necessary **Feature name** from the list.
   Features define a list of possible matching tokens. Features are collections of words or other features that can be categorized or grouped pragmatically, semantically, or syntactically.
   Extending the dictionaries in a knowledge base can improve the effectiveness of your text analysis, without requiring you to modify the knowledge base itself. For example, you if a new medication is released to the public, you can add the name of the medication to your medicines dictionary

10. Select the necessary result set with the help of ellipsis (...) in the **Result set** field. You can use a local file or a repository object as a result set.

11. Specify the column from which the words for the dictionary will be selected from the **Source column** list.

12. In the **Preview** area you see the first 50 items of the added dictionary.

13. Close the editor and save the created configuration to a file.

**Results**

You have a customized Knowledge Base configuration. Now you can begin using the Text Analytics feature.
Requirements for adding extensions to the knowledge base

You can add extensions to the knowledge base dictionaries. Adding extensions can increase the scope of entity extraction performed by Text Analytics feature.

A dictionary is a list of words or word combinations, usually grouped by different topics.

If you add extensions the knowledge base dictionary, the extensions must meet the following specifications:

- An extension must not begin with a numeral
- All extensions must begin on a new line
- All symbols other than letters, such as punctuation marks and separate numerals, must be enclosed in double quotation marks (" ").

Latin letters and numerals are permitted in knowledge base dictionaries.

Creating Simple Text Analytics projects

You can create a Simple Text Analytics project by using the New Simple Text Analytics Project wizard.

About this task

The Simple Text Analytics project serves as a container for documents to be analyzed by the Text Analytics feature. Unlike Text Analytics projects, Simple Text Analytics projects do not use a knowledge base and its configuration file. To analyze the text, you only need to indicate the type of elements that you want to look for and a dictionary containing the terms of the selected element type.

To create a Simple Text Analytics project:

Procedure

1. Open the New Simple Text Analytics Project wizard by selecting File > New > Other, expanding Text Analytics, selecting Simple Text Analytics Project and clicking Next.
2. Specify a name for the project in the Name field.
3. In the Sources to analyze area, you can double-click Folders or Result sets to specify a folder or result set objects that contains the data you want to analyze.

   Note: You will be able to specify additional folders from the Sources Explorer view or in the Edit Text Analytics project window.

4. You can also click Add source button to open New Input Source wizard:
5. If you want to add files from a local folder select System folder, click Next. The New Input Folder page opens.
   a. Browse the necessary directory with the help of ellipsis (...) button on your local disk.
   b. Select Include sub-folders check box analyze files in all sub-folders. Click Finish.
6. If you want to add a result set, select Result set from the list and click Next. The New Input Result Set page opens. Enter a path to the folder or use the ellipsis (...) to navigate to the folder. Click Next.
a. On the second page of New Input Result Set wizard specify the **ID column**, which is a unique result set row identifier.
b. Select the **Label column** from the list to specify the displaying name of the rows in the **Sources Explorer**.
c. Select **Source column** from the list to specify the source text for the analysis. If you select a binary column as a source, the **Encoding** field becomes available and you can select the necessary encoding for a column.
d. In the **Preview** area you see the first lines of the text from the added file which will be analyzed. Click **Finish**.

7. In the **Options** area select the **Wrap text** check box if you want each line to fit in the viewable editor window and allow the text to be read from top to bottom without any horizontal scrolling.

8. Select the **Save result set content files** check box to save the temporary result set files together with the project files. If this check box is cleared the result set files will be checked out every time when the project is opened.

9. Specify the required dictionaries on the **Dictionaries** page of the New Simple Text Analytics Project wizard.
   a. Select the **Include predefined dictionaries** check box to use built-in dictionaries such as names, e-mails, and telephone numbers in the analysis process.
   b. Add your own dictionaries in the New Dictionary wizard. To open the wizard, click **Add dictionary**.
   c. Once you have specified the dictionaries, you can export the setting to the file system. The next time you create a Simple Text Analytics project, you can import the saved settings and eliminate the need to re-create the list of dictionaries.

10. Select the **Run immediately** check box to run the analysis of the current project immediately.

11. Click **Finish** to close the New Simple Text Analytics Project wizard.

    **Note:** When the wizard closes you are prompted to switch to **Text Analytics** perspective if it is not opened yet. Select the **Remember my decision** check box to remember your choice and prevent opening the Switch perspective window again.

### Adding dictionaries to Simple Text Analytics projects

You can add your own dictionaries to a Simple Text Analytics project.

**About this task**

You can use any document that contains a result set as the source for the dictionary. For example, the source can be a simple text file, XML file, Query file, or Database file.

To add your own dictionary, perform the following procedure:

**Procedure**

1. On the **Dictionaries** page of the New Simple Text Analytics Project wizard, click **Add dictionary** to open the New Dictionary wizard.
2. In the **Dictionary** field, specify the name of the dictionary that will be displayed in the **Entities Explorer** tree.
3. In the **Result set** field, specify the path to the source of the dictionary. Click the ellipsis (…) to open the Result Set Source window, and select the required source from file or repository.

4. Select the column with required information from the **Source column** list. If you selected text file or another source with one column, it is selected automatically.

5. Click **Finish** to close the New Dictionary wizard. All the specified dictionaries are displayed in the **User dictionaries** table on the **Dictionaries** page of the New Simple Text Analytics Project wizard.

---

### Creating Text Analytics projects

You can create a Text Analytics project using the New Text Analytics Project wizard. The project serves as a container for documents to be analyzed by the Text Analytics feature.

#### Before you begin

Before creating a Text Analytics project, you must create a knowledge base configuration file.

#### About this task

To create a Text Analytics project:

#### Procedure

1. Open the New Text Analytics Project wizard in one of the following ways:
   - Click the **New Text Analytics Project** button ( ) on the toolbar.
   - From the **Text Analytics** perspective, click **File > New > Text Analytics Project**.
   - Select **File > New > Other** to open the New wizard. From the **DataQuant Objects** folder select **Text Analytics Project**.
   - From the **Text Analytics** perspective, click **New Text Analytics Project** button ( ) on the **Text Analytics Explorer** view.

2. Specify the Text Analytics project name in the **Name** field.

3. In the **Sources to analyze** area, you can double-click **Folders** or **Result sets** to specify a folder or result set objects that contains the data you want to analyze.

   **Note:** You will be able to specify additional folders from the **Sources Explorer** view or in the Edit Text Analytics project window.

4. You can also click **Add source** button to open New Input Source wizard:

5. If you want to add files from a local folder select **System folder**, click **Next**. The New Input Folder page opens.
   - Browse the necessary directory with the help of ellipsis (…) button on your local disk.
   - Select **Include sub-folders** check box analyze files in all sub-folders. Click **Finish**.

6. If you want to add a result set, select **Result set** from the list and click **Next**. The New Input Result Set page opens. Enter a path to the folder or use the ellipsis (…) to navigate to the folder. Click **Next**.
a. On the second page of New Input Result Set wizard specify the ID column, which is a unique result set row identifier.

b. Select the Label column from the list to specify the displaying name of the rows in the Sources Explorer.

c. Select Source column from the list to specify the source text for the analysis. If you select a binary column as a source, the Encoding field becomes available and you can select the necessary encoding for a column.

d. In the Preview area you see the first lines of the text from the added file which will be analyzed. Click Finish.

7. In the Options area select the Wrap text check box if you want each line to fit in the viewable editor window and allow the text to be read from top to bottom without any horizontal scrolling.

8. Select the Save result set content files check box to save the temporary result set files together with the project files. If this check box is cleared the result set files will be checked out every time when the project is opened.

9. In the New Input Source wizard click Next.

   In the Knowledge Base configuration window, specify the path to the knowledge base configuration file in the Configuration field.

   When specified, the configuration file provides metadata for Text Analytics project, including the lists of available loads and processing sequences.

   If this metadata is imported successfully, the Load and Processing sequence lists become available.

   Note: You can click Browse to locate the knowledge base configuration file.

10. From the Load list, select the required load that is used for the analysis. A named setting that determines how documents will be loaded into Text Analyzer.

11. From the Processing sequence list, select the required knowledge base processing sequence. This sequence of actions will be performed to every text file under analysis.

12. Select the Run immediately check box to run the analysis of the current project immediately.

13. Click Finish to close the New Text Analytics Project wizard.

   Note: When the wizard closes you are prompted to switch to Text Analytics perspective if it is not opened yet. Select the Remember my decision check box to remember your choice and prevent opening the Switch perspective window again.

---

**Editing Text Analytics project**

You can edit a Text Analytics project using the Edit Text Analytics Project window.

**About this task**

To edit a Text Analytics project:

**Procedure**

1. From the Text Analytics perspective, right-click the required Text Analytics project in the Text Analytics Explorer view, and then select Edit from the pop-up menu.
2. In the **Sources to analyze** list, specify the sources that contain files to be analyzed. You can add and remove folders from the list by clicking **Add source** and **Remove source**.

**Note:** If you remove a source from the project, the analysis results of files contained in this source are also removed from the **Entities Explorer** view.

3. You can also modify source files by clicking **Edit source**. In the Edit Text Analytics Project dialog specify the necessary parameters and click **OK**.

4. Select **Wrap text** check box if you want each line to fit in the viewable editor window, allowing text to be read from top to bottom without any horizontal scrolling.

5. Select **Save result set content files** check box to save the temporary result set files with the project files. If this check box is cleared the result set files will be checked out every time when the project is opened.

6. Specify the path to the knowledge base configuration file in the **Configuration** field on the **Knowledge Base setting** page.

**Note:** You can click **Browse** to locate the knowledge base configuration file.

7. From the **Load** list, select the required load that is used for the analysis. A named setting that determines how documents will be loaded into Text Analyzer.

8. From the **Processing sequence** list, select the required knowledge base processing sequence. This sequence of actions will be performed to every text file under analysis.

9. Click **OK** to close the Edit Text Analytics Project window.

Depending on the changes you apply to the Text Analytics project, you might be prompted to repeat the analysis of the text documents. If you have added a folder to the project, you are prompted to analyze the files contained in this folder. If you have changed the knowledge base settings, you are prompted to re-execute the project.

---

**Analyzing documents in a Text Analytics project**

You use the Text Analytics projects to analyze unstructured text documents and reconcile the necessary information cited across these documents.

**About this task**

The work with Text Analytics project involves the usage of the **Text Analytics** perspective that provides four views and a text viewer.

To analyze documents in a Text Analytics project:

**Procedure**

1. Run the Text Analytics project that you have created.

   To do this, select the required project from the **Text Analytics Explorer** view, and then click **Run** toolbar button in the **Text Analytics Explorer** view.

   - If you have opened an existing project, you do not have to run it again unless its structure has been edited since the last execution time. The files that have been already analyzed are marked with a green dot ( ).
   - If the file has been modified after the last analysis, it is marked with a yellow dot ( ).
• If the file has been added after last running the project, it is marked with a red dot ( ). All the project files are marked with red in case after its last analysis the knowledge base settings have been modified.

2. Use the Entities Explorer view to observe the analysis results tree. The nodes of this tree represent structure of entities that have been found and used in the analyzed files.

An entity is a representation of people, places, organizations, and time expressions found in a text document. They are grouped / categorized by shared characteristics and are analyzed grammatically, semantically, or pragmatically. The entities refer to feature that can be extended by adding terms to them.

The number of unique elements found in these files according to a specific entity is show in brackets after the name of this entity.

3. You can study the graphical representation of word usage statistic with the help of the Word Map view. To do this, double-click the required entity in the Entities Explorer view. The Word Map displays the words and word combinations found in analyzed files on the basis of the selected entity.

4. Use the Word count slider to change the number of words and word combinations the Word Map displays. The value can vary from 1 to maximum value that you have specified in the Word Map Setting window.

Note: The default maximum value is set to 100.

5. If you want to continue working with any of displayed words or phrases, select the required element on the Word Map. The specified element is highlighted with black color. The Sources Explorer view displays only those files that contain the specified element.

Note: You can search for specific element displayed on the Word Map, by typing it in the Search field.

6. Use Previous and Next buttons to page the files of a particular source if there are more than 40 of them in it.

7. In the Sources Explorer view, double-click the file that you want to observe. The file opens in the Highlighter at the position of the first occurrence of the word or phrase that you have selected in the Word Map view. Every instance of this selected element is highlighted.

8. Use the navigation buttons on the toolbar to scroll the document to the positions of first, previous, next, or last occurrence of the selected element.

9. Select Wrap text check box on the toolbar if you want each line to fit in the viewable editor window, allowing text to be read from top to bottom without any horizontal scrolling. If this check box is cleared the text in the Highlighter will have its original formatting.

---

**Specifying Word Map view settings**

You can modify Word Map view settings to change the representation of the analysis results in the view.

**About this task**

The Word Map displays the text analytics results in the form of a word cloud, where the level of importance and relevance of each word is related to the font size applied to the word.
To specify Word Map view settings:

**Procedure**

1. Click the Word Map settings icon ( ) in the Word Map view. Word Map settings dialog opens.
2. Select the rotation from the Rotation list. The available rotations are:
   - **Horizontal** sets a horizontal rotation of the words in the cloud
   - **Vertical** sets a vertical rotation of the words in the cloud.
   - **Both** sets both horizontal and vertical rotations of the words in the cloud.
   - **Any** sets an arbitrary rotation of the words in the cloud.
3. From the Color list select the color palette for the word cloud. The available palettes are:
   - **Random RGB** sets a wide range of colors and their shades.
   - **Random Palette** sets 8 preset colors.
4. Select the font from the Font list of the available ones.
5. Set the maximum words amount to be displayed in the view by specifying it in the Max words field. The default value is 100.
6. Select the Compact check box to represent the word in the cloud more neatly.

   **Note:** It takes more time to generate a compact word map.
7. Click Defaults to restore the default settings values.
8. Click OK to save the changes.
9. Click the Generate icon ( ) in the Word Map view to apply the settings and refresh Word Map view.

**Exporting Word Map results**

You can export the results of the analysis from the Word Map view to a file or database.

**About this task**

To export the analysis results:

**Procedure**

1. Click the Export Word Map ( ) icon from the toolbar in the Word Map view.
2. Select Use word count filter check box to limit the amount of the exported words by the value of Word count slider.
3. In the Export content select the button that corresponds to what you want to export.
   a. Select Export image to export the results as an image to a file.
      1) In the Path field, select the necessary path to save the image.
      2) From the Format list, select the necessary image format and click Finish.
      The image will be exported to the specified file.
   b. Select Export results to export the words and their weights to file or database.
1) Click **File system** to export the Word Map results data to a file. The fields that are required to export Word Map results to a file system are displayed.

2) Click **Database** to export the Word Map results data to a database. The fields that are required to export Word Map results to a database are displayed.

**Note:** The steps for Exporting Word Map results to a file or database are the same as exporting query results to a file or database. See [Exporting query results to a database](#) and [Exporting query results to a file](#) for a description of the steps.
Chapter 16. Working with global variables

Global variables are variables that stay active while the current session of DataQuant is active. This is in contrast to substitution variables that are active only during the execution of an object (query, form, procedure).

For objects that use substitution variables, values are entered via a window when the object is run. After the object runs, the variable and the value that you used no longer exists. For objects that use global variables, the value currently defined for the global variable is used.

There are two types of global variables:
- **System global variables** are pre-loaded with the DataQuant application. That means their values are re-initialized each time you open the DataQuant application. System global variables begin with the letters DSQQW, DSQAO, DSQEC, DSQDC, and DSQCP. You cannot add or delete system global variables. However, you can edit the default values for the system global variables.
- **User global variables** are defined by the user. User global variables can be specified with any unique name that does not begin with the letters DSQQW, DSQAO, DSQEC, DSQDC, and DSQCP. User global variables values are not saved when you terminate the DataQuant session.

Using the Global Variables page of the Preferences window you can view, add, delete and edit your user global variables. You can also edit the values of the system global variables.

Specifying user global variables

You use the Global Variables page of the Preferences window to view, add, delete and edit your user global variables.

About this task

To view, add, edit and delete user defined global variables:

Procedure

2. The system and user global variables that are defined for your session are listed in the Variables list box. The name of the global variable is listed in the Name field. The current value for the global variable is listed in the Value field.
3. To add a new user defined global variable, click the Add button. The New variable window opens. Type the name of the new user global variable in the Variable name field. You can type any combination of unique characters that have not already been used. You can not begin a user global variable with the system global variable prefixes of DSQQW, DSQAO, DSQEC, DSQDC, and DSQCP. Click OK. The user global variable is added to the list and will be available for use during the current DataQuant session. To specify the initial value of the variable, edit the Value cell as described in the following step.
4. To edit the value of a user global variable, click in the **Value** field for the global variable that you want to change. Type the new value for the global variable in the **Value** field.

5. To delete a user global variable, select an existing user global variable from the **Variables** list and click the **Remove** button. The user global variable is deleted.

6. Click **Apply** to save your preferences choices.

7. Click **Restore Defaults** to restore the default preference values.

8. Click **OK** to close the Preferences dialog.

---

**Editing system global variable values**

You use the Global Variables page of the Preferences dialog to edit any system global variable values.

**About this task**

To edit system global variable values:

**Procedure**


2. The system global variables are listed in the **Variables** list by name. System global variable start with the prefix of **DSQQW**, **DSQAO**, **DSQEC**, **DSQDC**, or **DSQCP**.

3. The current system global variable values are listed in the **Value** field. To edit the value of a global variable, click in the **Value** field for the system global variable that you want to change. Type the new value for the system global variable in the **Value** field.

4. Click **Restore Defaults** to restore the default values for all the system global variables.

5. Click **Apply** to save your preferences choices.

6. Click **OK** to close the Preferences window.

---

**System defined global variables**

System defined global variables are pre-loaded with the DataQuant application. Their values are re-initialized each time you open the application.

The system defined global variables are divided into the following groups:

**DSQQW global variables**

Global variables whose names begin with **DSQQW** provide information about the current query environment.

The following DSQQW global variables are currently available:

<table>
<thead>
<tr>
<th>Name</th>
<th>Length</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DSQQW_AUTOMATION</td>
<td>1</td>
<td>Indicates whether the application was started as an Automation server.</td>
</tr>
<tr>
<td>Name</td>
<td>Length</td>
<td>Description</td>
</tr>
<tr>
<td>-------------------</td>
<td>--------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>DSQQW_CONNECTIONS</td>
<td>1</td>
<td>Controls the use of database server connections while running a procedure. Value can be zero (0) to minimize the number of connections or one (1) to allow a new connection for each RUN QUERY command. Specifying a value of zero (0) can force the distributed product to reset or complete a data object before continuing execution of a procedure. The default value is zero (0).</td>
</tr>
<tr>
<td>DSQQW_DQ</td>
<td>1</td>
<td>The value of the double quotation mark character. This variable can be used in queries and procedures to eliminate the need for the user to enter quotes with a text value. The default value is the double quotation mark character.</td>
</tr>
</tbody>
</table>
| DSQQW_EXP_DT_FRMT | 1      | The format to use when exporting data with the EXPORT DATA command in a procedure. Specify the value of:  
- zero (0) for text format  
- two (2) for HTML format  
- three (3) for CSV format  
- four (4) for IXF format  
- five (5) for dBASE III files  
- six (6) for Shapefile format  
- seven (7) for WQML format  
- eight (8) for XML format  
- nine (9) for PDF format  
- ten (10) for XLS format  
- eleven (11) for XLSX format |
<p>| DSQQW_EXP_OUT_MDE | 1      | The IXF variation to use when exporting data to an IXF file. Value can be zero (0) for System/370 character-mode IXF or one (1) for PC/IXF. The default value is one (1). |
| DSQQW_FST_SV_DATA | 1      | Controls the use of “fast mode” when saving data with the SAVE DATA command in a procedure. Value can be zero (0) to use regular save mode (not fast mode); one (1) to use fast mode with ORDER BY clause(s) stripped; or two (2) to use fast mode with ORDER BY clause(s). The default value is zero (0). For more information on saving data using fast mode see “Saving data using the regular and fast save methods” on page 136. |</p>
<table>
<thead>
<tr>
<th>Name</th>
<th>Length</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DSQQW_HTML_REFTEXT</td>
<td>55</td>
<td>The text that appears in a report when the &amp;REF variable is used. The default value is &quot;Back To&quot;.</td>
</tr>
<tr>
<td>DSQQW_ORIENTATION</td>
<td>0</td>
<td>The orientation of the application. The value is zero (0) for left-to-right orientation. The value is one (1) for right-to-left orientation.</td>
</tr>
<tr>
<td>DSQQW_PROC_FAIL_ON_ERROR</td>
<td>1</td>
<td>Stops procedure execution if any of the procedure commands fails. A value of zero (0) specifies the procedure will continue. A value of one (1) specifies the procedure will stop.</td>
</tr>
<tr>
<td>DSQQW_PROC_OUTPUT</td>
<td></td>
<td>Output file name for a procedure.</td>
</tr>
<tr>
<td>DSQQW_PROC_WNDWS</td>
<td>1</td>
<td>Controls what happens to intermediate result windows created by running a procedure. The value of zero (0) will close all intermediate windows, leaving only the final result window open at the end of the procedure. The value of one (1) will leave all windows open at the end of the procedure. The value of two (2) will close all intermediate windows, and will also close the procedure window if the procedure is run indirectly (run from another procedure or from the command line). The default value is one (1).</td>
</tr>
<tr>
<td>DSQQW_QUERY_LANG</td>
<td>1</td>
<td>Specifies the subtype of query created when a DISPLAY QUERY command is executed but no query object exists. Value can be zero (0) for a query in the SQL view or one (1) for a query in the prompted view. The default value is zero (0).</td>
</tr>
<tr>
<td>DSQQW_QUERY_PREP</td>
<td>1</td>
<td>Specifies whether the query on a RUN command is to be prepared or run. The results of prepared queries are not returned to the user's workstation. Value can be zero (0) to prepare the query, or one (1) to run the query. The default value is one (1).</td>
</tr>
<tr>
<td>DSQQW_QUERY_PRESERVE_SORT</td>
<td>1</td>
<td>Specifies whether the query sorting order defined by a user is saved within the query and used every time the query is run. Value can be zero (0) - not to preserve the sorting order, or one (1) - to preserve the sorting order. The default value is one (1).</td>
</tr>
<tr>
<td>Name</td>
<td>Length</td>
<td>Description</td>
</tr>
<tr>
<td>----------------------</td>
<td>--------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>DSQQW_REUSE_OBJS</td>
<td>1</td>
<td>Specifies whether existing windows displaying retrieved objects are reused, or if a new window opens every time an object is selected. Value can be zero (0) to always open objects in new windows, or (1) to activate an existing window if the selected object is already open. The default value is one (1).</td>
</tr>
<tr>
<td>DSQQW_RPT_COPIES</td>
<td>10</td>
<td>Specifies the number of copies to print when printing a report with the PRINT REPORT command in a procedure. The default value is one (1).</td>
</tr>
<tr>
<td>DSQQW_RPT_FONT</td>
<td>55</td>
<td>Specifies the font face name to use when printing a report with the PRINT REPORT command in a procedure. The default value is &quot;Monospaced&quot;.</td>
</tr>
<tr>
<td>DSQQW_RPT_FONT_BD</td>
<td>1</td>
<td>Specifies the font bold attribute to use when printing a report with the PRINT REPORT command in a procedure. A value of zero (0) specifies not bold and a value of one (1) specifies bold. The default value is zero (0).</td>
</tr>
<tr>
<td>DSQQW_RPT_FONT_CS</td>
<td>3</td>
<td>The character set of the font to use when printing a report with the PRINT REPORT command in a procedure. The default value is zero (0).</td>
</tr>
<tr>
<td>DSQQW_RPT_FONT_IT</td>
<td>1</td>
<td>Specifies the font italic attribute to use when printing a report with the PRINT REPORT command in a procedure. A value of zero (0) specifies not italic and a value of one (1) specifies italic. The default value is zero (0).</td>
</tr>
<tr>
<td>DSQQW_RPT_FONT_SZ</td>
<td>2</td>
<td>Specifies the font point size to use when printing a report with the PRINT REPORT command in a procedure. The default value is ten (10).</td>
</tr>
<tr>
<td>DSQQW_RPT_LEN_TYP</td>
<td>1</td>
<td>Specifies the type of page length that will be used when printing a report with the PRINT REPORT command or exporting a report with the EXPORT REPORT command in a procedure. Value can be zero (0) to automatically fit the length to the printed page, one (1) to specify an explicit number of lines, or two (2) to specify a continuous report with no page breaks. The default value is zero (0).</td>
</tr>
<tr>
<td>Name</td>
<td>Length</td>
<td>Description</td>
</tr>
<tr>
<td>--------------------</td>
<td>--------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>DSQQW_RPT_NUM_CHR</td>
<td>10</td>
<td>Specifies the number of characters to fit across a printed page when printing a report with the PRINT REPORT command or exporting a report with the EXPORT REPORT command in a procedure. This has an effect only when DSQQW_RPT_WID_TYP is one (1). The default value is eighty (80).</td>
</tr>
<tr>
<td>DSQQW_RPT_NUM_LNS</td>
<td>10</td>
<td>Specifies the number of lines to fit down a printed page when printing a report with the PRINT REPORT command or exporting a report with the EXPORT REPORT command in a procedure. This has an effect only when DSQQW_RPT_LEN_TYP is one (1). The default value is sixty (60).</td>
</tr>
<tr>
<td>DSQQW_RPT_ORIENT</td>
<td>1</td>
<td>The page orientation to use when printing a report with the PRINT REPORT command or exporting a report with the EXPORT REPORT command in a procedure. Value can be zero (0) for portrait or one (1) for landscape. The default value is zero (0).</td>
</tr>
<tr>
<td>DSQQW_RPT_OUT_TYP</td>
<td>1</td>
<td>The format to use when printing a report with the PRINT REPORT command in a procedure. Value can be zero (0) for text or two (2) for HTML. The default value is zero (0).</td>
</tr>
<tr>
<td>DSQQW_RPT_TD_TYP</td>
<td>1</td>
<td>Date format for TD edit code. Value can be zero (0) for ISO format, one (1) for USA format, two (2) for EUR format or three (3) for JIS format. The default value is zero (0).</td>
</tr>
<tr>
<td>DSQQW_RPT_TT_TYP</td>
<td>1</td>
<td>Time format for TT edit code. Value can be zero (0) for ISO format, one (1) for USA format, two (2) for EUR format or three (3) for JIS format. The default value is zero (0).</td>
</tr>
<tr>
<td>DSQQW_RPT_USE_PS</td>
<td>1</td>
<td>Specifies what page formatting options (page length, page width, etc.) to use when printing a report with the PRINT REPORT command in a procedure. Value can be zero (0) to use the values specified on the PRINT REPORT command or in global variables, or one (1) to use the values specified in the form's page setup. The default value is one (1).</td>
</tr>
<tr>
<td>Name</td>
<td>Length</td>
<td>Description</td>
</tr>
<tr>
<td>----------------------</td>
<td>--------</td>
<td>--------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>DSQQW_RPT_WID_TYP</td>
<td>1</td>
<td>Specifies the type of page width when printing a report with the PRINT REPORT command in a procedure. Value can be zero (0) to automatically fit the width to the printed page, one (1) to specify an explicit number of characters or two (2) to specify a continuous line. The default value is zero (0).</td>
</tr>
<tr>
<td>DSQQW_SHOW_QUERY</td>
<td>1</td>
<td>Specifies which view of a query to display when a SHOW QUERY command is issued from a procedure. Value can be zero (0) for SQL or prompted view, one (1) for results view, two (2) for build view, or three (3) for layout view. The default value is zero (0).</td>
</tr>
<tr>
<td>DSQQW_SQ</td>
<td>1</td>
<td>The value of a single quote character. This variable can be used in queries and procedures to eliminate the need for the user to enter quotes with a text value. The default value is a single quote character (&quot;).</td>
</tr>
<tr>
<td>DSQQW_SV_DATA_C_S</td>
<td>10</td>
<td>The number of rows to insert before committing the unit of work when saving data with a SAVE DATA command in a procedure. Value can be zero (0) for all of the rows or an explicit number of rows. The default value is zero (0).</td>
</tr>
<tr>
<td>DSQQW_SV_DATA_T_M</td>
<td>1</td>
<td>Specifies how source and target column data types are matched when using the SAVE DATA command. The value can be zero (0) to require exact data type matches, one (1) to allow data type conversions with no possible data loss, or two (2) to allow all data type conversions that are supported by the database. The default value is one (1).</td>
</tr>
<tr>
<td>DSQQW_UEDIT_JAR</td>
<td>55</td>
<td>The name of the JAVA archive file that contains user edit routines.</td>
</tr>
</tbody>
</table>

**DSQAO global variables**

Global variables whose names begin with DSQAO provide information about the current state of the query session.

The following DSQAO global variables are available:
<table>
<thead>
<tr>
<th>Name</th>
<th>Length</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DSQAO_BATCH</td>
<td>1</td>
<td>Batch or interactive mode. Value can be one (1) for an interactive session or two (2) for a batch session. See the BATCH command line parameter.</td>
</tr>
<tr>
<td>DSQAO_CONNECT_ID</td>
<td>8</td>
<td>The user ID that is used to connect to the current database.</td>
</tr>
<tr>
<td>DSQAO_CURSOR_OPEN</td>
<td>1</td>
<td>The status of the current query object's database cursor. Value can be one (1) if the cursor is open or two (2) if the cursor is closed.</td>
</tr>
<tr>
<td>DSQAO_DBCS</td>
<td>1</td>
<td>DBCS support status. Value can be one (1) if DBCS support is present or two (2) if DBCS support is not present.</td>
</tr>
<tr>
<td>DSQAO_HOME_WORKSPACE</td>
<td>128</td>
<td>The current repository user's home workspace key, if the workspace exists. Valid values are:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• rsbi:/.workspaces/&lt;user name&gt;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>This is the value if the user connected to a secured repository connection and if the rsbi:/.workspaces/&lt;user name&gt; object is viewable by the current user in the repository. &lt;user name&gt; is the login name of the repository user.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• rsbi:/.workspaces</td>
</tr>
<tr>
<td></td>
<td></td>
<td>This is the value if the user connected to a repository connection without security, or the rsbi:/.workspaces/&lt;user name&gt; is not viewable by the user, or it does not exist.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• blank</td>
</tr>
<tr>
<td></td>
<td></td>
<td>This is value if the previous situations do not exist. For example, if the user is not connected to any repository connection.</td>
</tr>
<tr>
<td>DSQAO_NLF_LANG</td>
<td>1</td>
<td>National language of session. Value is &quot;E&quot; for the English language.</td>
</tr>
<tr>
<td>DSQAO_NUM_FETCHED</td>
<td>0</td>
<td>The number of rows fetched by the current query object.</td>
</tr>
<tr>
<td>DSQAO_OBJ_NAME</td>
<td>18</td>
<td>The name of the current query, form, or procedure object. If there is no current object, the value is blank.</td>
</tr>
<tr>
<td>DSQAO_OBJ_OWNER</td>
<td>8</td>
<td>The owner of the current query, form, or procedure object. If there is no current object, the value is blank.</td>
</tr>
<tr>
<td>DSQAO_REP_USER</td>
<td>8</td>
<td>The user name that is used to connect to the current repository.</td>
</tr>
<tr>
<td>DSQAO_RSB1_RELEASE</td>
<td>2</td>
<td>Numeric release number of the application.</td>
</tr>
<tr>
<td>DSQAO_RSB1_VER_RLS</td>
<td>10</td>
<td>External version and release number for the application.</td>
</tr>
</tbody>
</table>

Note: Workspace operations such as creating, deleting, and renaming performed by the current user affect the Global Variable value. Additionally, such operations performed by other users might also affect the value.
<table>
<thead>
<tr>
<th>Name</th>
<th>Length</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DSQAO_QUERY_MODEL</td>
<td>1</td>
<td>Model of the current query object. Value can be one (1) for relational.</td>
</tr>
<tr>
<td>DSQAO_QRY_SUBTYPE</td>
<td>1</td>
<td>Subtype of the current query object. Value can be one (1) for SQL queries or three (3) for queries in a prompted view.</td>
</tr>
<tr>
<td>DSQAO_SYSTEM_ID</td>
<td>1</td>
<td>Current® operating system. Values can be one of the following:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 8 - Windows NT and above</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 9 - Linux</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 10 - HP-UX</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 11 - AIX®</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 12 - Solaris</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 13 - iSeries</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 14 - z/OS</td>
</tr>
</tbody>
</table>

**DSQEC global variables**

Global variables whose names begin with DSQEC control how commands and procedures are executed.

The following DSQEC global variables are available:

<table>
<thead>
<tr>
<th>Name</th>
<th>Length</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DSQEC_CON_ACC_RES</td>
<td>1</td>
<td>For executable SELECT queries that the application submits to DB2 for z/OS, this variable allows you to specify how you want the database to proceed when the data to be selected is locked by an insert, update, or delete operation. When you set this variable, the application specifies the clause associated with the variable value on the concurrent-access-resolution attribute of the PREPARE statement for the SELECT query. Executable SELECT queries can result not only from queries (such as SQL SELECT queries, prompted queries, or QBE P. queries), but also from other operations such as DISPLAY TABLE. Possible values are:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 0 - No concurrent access resolution options on the PREPARE statement associated with the pending SQL SELECT statement are specified. This is the default value.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 1 - SKIP LOCKED DATA. This value can be specified for executable SELECT statements directed to DB2 for z/OS Version 9 or later.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 2 - USE CURRENTLY COMMITTED. This value can be specified for executable SELECT statements directed to DB2 for z/OS Version 10 and later.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 3 - WAIT FOR OUTCOME. This value can be specified for executable SELECT statements directed to DB2 for z/OS Version 10 and later.</td>
</tr>
<tr>
<td>Name</td>
<td>Length</td>
<td>Description</td>
</tr>
<tr>
<td>------------------</td>
<td>--------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>DSQEC_EXTEND_STG</td>
<td>31</td>
<td>Specifies the number of megabytes of extended storage that the application will acquire on each request to the extended storage manager when spilling data to extended storage in QMF for TSO. When a user performs an operation that requires extended storage, the application issues repeated requests to the extended storage manager for the specified amount until the operation is complete or extended storage is exhausted. When setting this global variable, consider the average size of DATA objects with which your users work. If the average size is very large and you set the DSQEC_EXTEND_STG variable too low, the application must issue many calls to the extended storage manager to complete the DATA object, which could affect overall performance. Values can be from 1 to 1000. The default value is 25, indicating that the application requests 25MB of storage on each request.</td>
</tr>
<tr>
<td>DSQEC_FORM_LANG</td>
<td>1</td>
<td>Defines the default NLF language in which a form will be saved or exported. Value can be zero (0) for the presiding NLF language or one (1) for English. The default value is one (1).</td>
</tr>
</tbody>
</table>
| DSQEC_LAST_RUN   | 1      | Specifies the set of commands that cause the LAST_USED column of the Q.OBJECT_DIRECTORY table to be updated. Possible values are:  
• 0 - Last used is updated on any activity.  
• 1 - Last used is updated when RUN, SAVE, or IMPORT commands are performed. |
| DSQEC_LIST_OWNER | 128    | Provides the default value for the OWNER parameter of the LIST command. Specify an authorization ID up to 128 characters long. This variable is blank by default, resulting in a list of objects owned by the current authorization ID. You can use selection symbols in the variable value. Use an underscore (_) in place of a single character and a percent sign (%) in place of zero or more characters. For example, after you issue the following command, followed by a LIST command, the application lists only objects that are owned by user IDs that begin with the characters RO:  
```
SET GLOBAL (DSQEC_LIST_OWNER=RO%
```

The following command sets the default owner to any user IDs that begin with I, have any character in the second position, and any characters in the remaining positions:  
```
SET GLOBAL (DSQEC_LIST_OWNER=I_%
```

<p>| DSQEC_NLFCMD_LANG | 1      | Defines the expected NLF language for commands in procedures. Value can be zero (0) for the presiding NLF language or one (1) for English. The default value is zero (0). |
| DSQEC_RESET_RPT   | 1      | Determines whether a user will be prompted when an incomplete data object that will affect performance is encountered. Value can be zero (0) to complete the data object without prompting, one (1) to prompt the user asking whether the data object should be completed, or two (2) to reset the data object without prompting. |</p>
<table>
<thead>
<tr>
<th>Name</th>
<th>Length</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DSQEC_RUN_MQ</td>
<td>1</td>
<td>Specifies whether the RUN QUERY command supports multiple statements in an SQL query. Possible values are:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 0 - Multiple SQL statements are not supported.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>If you set this variable to 0 and run an SQL query that contains multiple statements, the application ignores all statements after the first semicolon.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 1 - Multiple SQL statements are supported.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>This is the default value. A semicolon can be placed at the end of each statement. You can substitute the semicolon with any other character by using SET STATEMENT DELIMITER comment at the beginning of the SQL text. For example, the following example is a valid use of SQL with multiple statements:</td>
</tr>
<tr>
<td></td>
<td></td>
<td><code>--SET STATEMENT DELIMITER=&quot;!&quot;</code></td>
</tr>
<tr>
<td></td>
<td></td>
<td><code>select * from q.staff!</code></td>
</tr>
<tr>
<td></td>
<td></td>
<td><code>select * from q.org</code></td>
</tr>
<tr>
<td>DSQEC_SHARE</td>
<td>1</td>
<td>Specifies the default value for whether a saved object will be shared with other users. Value can be zero (0) to not share the object or one (1) to share the object.</td>
</tr>
<tr>
<td>DSQEC_SP_RS_NUM</td>
<td>1</td>
<td>Specifies the number of the result set that will be displayed for a stored procedure. The default result set number is minus one (-1).</td>
</tr>
<tr>
<td>DSQEC_SQLQRYSZ_2M</td>
<td>1</td>
<td>Controls whether SQL queries greater than 32,767 bytes (32 KB) in length are supported by the RUN QUERY command. Possible values are:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 0 - SQL queries directed to DB2 for z/OS, iSeries, Linux and UNIX, and Windows databases are limited to 32,767 bytes (32 KB). This is the default.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 1 - SQL queries can be greater than 32 KB. The maximum supported query size varies depending on the type of database to which the query is directed:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>– Queries directed to DB2 for z/OS can be up to 2 MB in length.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>– Queries directed to DB2 for iSeries or DB2 for Linux, UNIX, and Windows can be up to 65 KB in length.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>These maximums assume that the version of the database to which the RUN QUERY command is directed supports queries of this size. SQL queries directed to DB2 for VM and VSE are limited to 8 KB.</td>
</tr>
</tbody>
</table>

**DSQDC global variables**

Global variables whose names begin with DSQDC control how information is displayed.

The following DSQDC global variables are available:
<table>
<thead>
<tr>
<th>Name</th>
<th>Length</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DSQDC_COL_LABELS</td>
<td>1</td>
<td>Specifies whether column headings will be column names or database labels in Classic Reports. Value can be zero (0) to specify that column headings will be column names or one (1) to specify that column headings will be database labels. The default value is one (1).</td>
</tr>
<tr>
<td>DSQDC_CURRENCY</td>
<td>18</td>
<td>Defines the custom currency symbol to use when the DC edit code is specified.</td>
</tr>
<tr>
<td>DSQDC_DISPLAY_RPT</td>
<td>1</td>
<td>Specifies whether a report is displayed after a RUN QUERY command in a procedure. Value can be zero (0) to not display a report or one (1) to automatically display a report with the default form. The default value is zero (0).</td>
</tr>
<tr>
<td>DSQDC_POS_SQLCODE</td>
<td>1</td>
<td>Specifies what happens when a positive SQL code is returned from the database. Possible values are:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 0 - Neither log the message or display the message text.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 1 - Log the message associated with the SQL code.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 2 - Display the online help that is associated with the SQL code.</td>
</tr>
</tbody>
</table>

**DSQCP global variables**

Global variables whose names begin with DSQCP control the operation of the table editor.

The following DSQCP global variables are available:

<table>
<thead>
<tr>
<th>Name</th>
<th>Length</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DSQCP_TEDFLT</td>
<td>1</td>
<td>Defines the reserved character used in the Table Editor to indicate a default value for a column. The default value is &quot;+&quot;. This variable can also be set in the Enter default values as field found in the Preferences window.</td>
</tr>
<tr>
<td>DSQCP_TENULL</td>
<td>1</td>
<td>Defines the reserved character used in the Table Editor to indicate a null value for a column. The default value is &quot;-&quot;. This variable can also be set in the Enter null values as field found in the Preferences window.</td>
</tr>
</tbody>
</table>
Chapter 17. Working with edit codes

Edit codes specify how to format the data that will be in a specific column of a report.

A default edit code that is based on the data type of the column is automatically assigned to each column in a report. You can change the default edit code for a column in a report from the Form dialog. The following types of edit codes are available:

Edit codes for character data

You use character data edit codes to format text fields in reports.

The following character data edit codes are available:

<table>
<thead>
<tr>
<th>Edit Code</th>
<th>Format</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>C</td>
<td>Display character data.</td>
<td>No special formatting.</td>
</tr>
<tr>
<td>CW</td>
<td>Display character data with wrapping based on column width.</td>
<td>No special formatting unless the value cannot fit on one line in the column. In that case, as much data as possible is placed within the column, and then additional data is wrapped to subsequent lines in the column.</td>
</tr>
<tr>
<td>CT</td>
<td>Display character data with wrapping based on the text in the column.</td>
<td>No special formatting unless the value cannot fit on one line in the column. In that case, as much data as possible is placed on a line within the column, until a blank is found within the text that triggers the wrapping of additional data to subsequent lines in the column. If the string of text is too long to fit in the column and does not contain a blank, the data is wrapped by width until a blank is found.</td>
</tr>
<tr>
<td>CDx</td>
<td>Display character data with wrapping based on the specified delimiter.</td>
<td>No special formatting unless the value cannot fit on one line in the column. In that case, a new line of data begins in the column each time a special delimiter is encountered in the text. If the string of text is too long to fit in the column and does not contain a delimiter, the data is wrapped by width until a delimiter is found. The delimiter specified by &quot;x&quot; can be any single character, including a blank. The delimiter character does not appear in the report.</td>
</tr>
<tr>
<td>X</td>
<td>Formats data as a series of hexadecimal characters.</td>
<td>No special formatting applied.</td>
</tr>
</tbody>
</table>
Edit codes for date data

You use date data edit codes to format fields containing date information in reports.

The letter "x" in the date edit codes represents the character to be used as the delimiter in the date value. The value of "x" can be any special character, including a blank, except a letter or a number. The following date data edit codes are available:

<table>
<thead>
<tr>
<th>Edit Code</th>
<th>Format</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>TDYx</td>
<td>YYYYxMMxDD</td>
<td>Year First</td>
</tr>
<tr>
<td>TDMx</td>
<td>MMxDDxYYYY</td>
<td>Month first</td>
</tr>
<tr>
<td>TDDx</td>
<td>DDxMMxYYYY</td>
<td>Day first</td>
</tr>
<tr>
<td>TYAx</td>
<td>YYxMMxDD</td>
<td>Two digit year first.</td>
</tr>
<tr>
<td>TDMAx</td>
<td>MMxDDxYY</td>
<td>Month first and two digit year.</td>
</tr>
<tr>
<td>TDDAx</td>
<td>DDxMMxYY</td>
<td>Day first and two digit year.</td>
</tr>
<tr>
<td>TDL</td>
<td>Locally defined</td>
<td>Formats date according to the format specified as the default at the database server requesting data.</td>
</tr>
<tr>
<td>TDT</td>
<td>Locally defined</td>
<td>Edit codes that appear on result set reports; that is, reports generated from a stored procedure CALL. They would be used on time or date data if the data was found not to be in ISO format. If these edit codes are found on column data, then the edit code cannot be changed for that column. Also, the report object cannot be exported if this edit code is present in the form.</td>
</tr>
</tbody>
</table>
Edit codes for graphic data

You use graphic data edit codes to format fields containing graphic or pure DBCS information in the reports.

The following graphic data edit codes are available:

<table>
<thead>
<tr>
<th>Edit Code</th>
<th>Format</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>G</td>
<td>Display graphic data</td>
<td>No special formatting.</td>
</tr>
<tr>
<td>GW</td>
<td>Display graphic data with wrapping based on column width.</td>
<td>No special formatting unless the value cannot fit on one line in the column. In that case, as much data as possible is placed within the column, and then additional data is wrapped to subsequent lines in the column.</td>
</tr>
</tbody>
</table>

Edit codes for numeric data

You use numeric data edit codes to format fields containing numeric information in reports.

The letters "nn" in the numeric data edit codes represents a number between 0 and 99. This number determines how many places to allow after the decimal point. The following numeric data edit codes are available:

<table>
<thead>
<tr>
<th>Edit Code</th>
<th>Format</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>E</td>
<td>Displays numbers in scientific notation.</td>
<td>Up to 17 significant digits, or up to 34 significant digits when editing extended floating point data, are shown even if the width of the column can accommodate more. Used as the default form for columns with data type FLOAT.</td>
</tr>
<tr>
<td>EZ</td>
<td>Displays numbers in scientific notation with zero values in the column suppressed.</td>
<td>Up to 17 significant digits, or up to 34 significant digits when editing extended floating point data, are shown even if the width of the column can accommodate more.</td>
</tr>
<tr>
<td>Dnn</td>
<td>Displays numbers in decimal notation formatted with a negative sign, thousands separator, and currency symbol.</td>
<td>The value &quot;nn&quot; is a number (0 to 99) that represents how many places to allow for the decimal point. Numbers with more places after the decimal are rounded and numbers with fewer places are padded.</td>
</tr>
<tr>
<td>DZnn</td>
<td>Displays numbers in decimal notation formatted with a negative sign, thousands separator, currency symbol and any zero values in the column suppressed.</td>
<td>The value &quot;nn&quot; is a number (0 to 99) that represents how many places to allow for the decimal point. Numbers with more places after the decimal are rounded and numbers with fewer places are padded.</td>
</tr>
<tr>
<td>Edit Code</td>
<td>Format</td>
<td>Notes</td>
</tr>
<tr>
<td>-----------</td>
<td>--------</td>
<td>-------</td>
</tr>
</tbody>
</table>
| DCn
n | Displays numbers in decimal notation formatted with a negative sign, thousands separator, and a user-defined currency symbol. | The value "nn" is a number (0 to 99) that represents how many places to allow for the decimal point. Numbers with more places after the decimal are rounded and numbers with fewer places are padded. The currency symbol that will be used instead of the standard currency symbol is defined using the global variable DSQDC_CURRENCY. |
<p>| DZCnn | Displays numbers in decimal notation formatted with a negative sign, thousands separator, a user-defined currency symbol and any zero values in the column suppressed. | The value “nn” is a number (0 to 99) that represents how many places to allow for the decimal point. Numbers with more places after the decimal are rounded and numbers with fewer places are padded. The currency symbol that will be used instead of the standard currency symbol is defined using the global variable DSQDC_CURRENCY. If both edit code options ”Z” and ”C” are used, ”C” must follow ”Z”. |
| Inn | Displays numbers in decimal notation formatted with leading zeros displayed and a negative sign. | The value ”nn” is a number (0 to 99) that represents how many places to allow for the decimal point. Numbers with more places after the decimal are rounded and numbers with fewer places are padded. |
| IZnn | Displays numbers in decimal notation formatted with leading zeros displayed, a negative sign and any zero values in the column suppressed. | The value ”nn” is a number (0 to 99) that represents how many places to allow for the decimal point. Numbers with more places after the decimal are rounded and numbers with fewer places are padded. |
| Jnn | Displays numbers in decimal notation formatted with leading zeros displayed. | The value “nn” is a number (0 to 99) that represents how many places to allow for the decimal point. Numbers with more places after the decimal are rounded and numbers with fewer places are padded. |
| JZnn | Displays numbers in decimal notation formatted with leading zeros displayed and any zero values in the column suppressed. | The value &quot;nn&quot; is a number (0 to 99) that represents how many places to allow for the decimal point. Numbers with more places after the decimal are rounded and numbers with fewer places are padded. |
| Knn | Displays numbers in decimal notation formatted with a negative sign and a thousands separator. | The value “nn” is a number (0 to 99) that represents how many places to allow for the decimal point. Numbers with more places after the decimal are rounded and numbers with fewer places are padded. |</p>
<table>
<thead>
<tr>
<th>Edit Code</th>
<th>Format</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>KZnn</td>
<td>Displays numbers in decimal notation formatted with a negative sign, a thousands separator and any zero values in the column suppressed.</td>
<td>The value &quot;nn&quot; is a number (0 to 99) that represents how many places to allow for the decimal point. Numbers with more places after the decimal are rounded and numbers with fewer places are padded.</td>
</tr>
<tr>
<td>Lnn</td>
<td>Displays numbers in decimal notation formatted with a negative sign.</td>
<td>The value &quot;nn&quot; is a number (0 to 99) that represents how many places to allow for the decimal point. Numbers with more places after the decimal are rounded and numbers with fewer places are padded.</td>
</tr>
<tr>
<td>LZnn</td>
<td>Displays numbers in decimal notation formatted with a negative sign and any zero values in the column suppressed.</td>
<td>The value &quot;nn&quot; is a number (0 to 99) that represents how many places to allow for the decimal point. Numbers with more places after the decimal are rounded and numbers with fewer places are padded.</td>
</tr>
<tr>
<td>Pnn</td>
<td>Displays numbers in decimal notation formatted with a negative sign, thousands separator, and a percent sign.</td>
<td>The value &quot;nn&quot; is a number (0 to 99) that represents how many places to allow for the decimal point. Numbers with more places after the decimal are rounded and numbers with fewer places are padded.</td>
</tr>
<tr>
<td>PZnn</td>
<td>Displays numbers in decimal notation formatted with a negative sign, thousands separator, a percent sign and any zero values in the column suppressed.</td>
<td>The value &quot;nn&quot; is a number (0 to 99) that represents how many places to allow for the decimal point. Numbers with more places after the decimal are rounded and numbers with fewer places are padded.</td>
</tr>
</tbody>
</table>

**Edit codes for time data**

You use time data edit codes to format fields containing time information in reports.

The letter "x" in the time data edit code represents the character that will be used as the delimiter in the time value. The value of "x" can be any character, including a space, except a letter or a number. The following time data edit codes are available:

<table>
<thead>
<tr>
<th>Edit Code</th>
<th>Format</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>TTSx</td>
<td>HHxMMxSS</td>
<td>24 hour clock, including seconds</td>
</tr>
<tr>
<td>TTCx</td>
<td>HHxMMxSS</td>
<td>12 hour clock, including seconds</td>
</tr>
<tr>
<td>TTAx</td>
<td>HHxMM</td>
<td>Abbreviated 24 hour clock (no seconds)</td>
</tr>
<tr>
<td>TTAN</td>
<td>HHMM</td>
<td>Abbreviated 24 hour clock (no seconds) without any delimiter between the time values.</td>
</tr>
<tr>
<td>TTUx</td>
<td>HHxMM PM HHxMM AM</td>
<td>USA format</td>
</tr>
<tr>
<td>Edit Code</td>
<td>Format</td>
<td>Notes</td>
</tr>
<tr>
<td>-----------</td>
<td>--------</td>
<td>-------</td>
</tr>
<tr>
<td>TTL</td>
<td>Locally defined.</td>
<td>Formats time data according to the format specified as the local default at the database server requesting data.</td>
</tr>
<tr>
<td>TTT</td>
<td>Locally defined</td>
<td>Edit codes that appear on result set reports; that is, reports generated from a stored procedure CALL. They would be used on time or date data if the data was found not to be in ISO format. If these edit codes are found on column data, then the edit code cannot be changed for that column. Also, the report object cannot be exported if this edit code is present in the form.</td>
</tr>
</tbody>
</table>

**Edit codes for timestamp data**

You use timestamp data edit codes to format fields containing timestamp information in reports.

The following timestamp edit codes are available:

<table>
<thead>
<tr>
<th>Edit Code</th>
<th>Format</th>
<th>Notes</th>
</tr>
</thead>
</table>
| TSI       | yyyy–mm–dd–hh.mm.ss.nnnnnnnnnnn | Where:  
• yyyy is the four digit year  
• mm is the two digit month  
• dd is the two digit day  
• hh is the two digit hour  
• mm is the two digit minute  
• ss is the two digit second  
• nnnnnnnnnnn is the twelve digit fractional seconds |
<table>
<thead>
<tr>
<th>Edit Code</th>
<th>Format</th>
<th>Notes</th>
</tr>
</thead>
</table>
| TSZ       | yyyy-mm-dd-hh.mm.ss.nnnnnnnnnn±th:tm | Where:  
• yyyy is the four digit year  
• mm is the two digit month  
• dd is the two digit day  
• hh is the two digit hour  
• mm is the two digit minute  
• ss is the two digit second  
• nnnnnnnnnnn is the twelve digit fractional seconds  
• ±th is the two-digit value representing the time zone hour, shown as an offset relative to UTC  
• tm is the two-digit value representing the time zone minutes between 0 and 59  
Note: To specify UTC, you can either specify a time zone of -24:00 or +24:00 or replace the time zone offset and its sign with an uppercase Z. |

**User defined edit codes**

You can use user defined edit codes Uxxxx and Vxxxx for special purposes.

The value "xxxx" can be any combination of characters, excluding embedded blanks.

The following user edit codes are predefined:

<table>
<thead>
<tr>
<th>Edit Code</th>
<th>Format</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>VSSN</td>
<td>xxx–xx–xxxx</td>
<td>Social security number format.</td>
</tr>
<tr>
<td>VTEL</td>
<td>(xxx) xxx–xxxx</td>
<td>Telephone number format.</td>
</tr>
<tr>
<td>VTEL2</td>
<td>xxx.xxx.xxx</td>
<td>Telephone number format.</td>
</tr>
<tr>
<td>VZIP</td>
<td>xxxxx–xxxx</td>
<td>Zip code format.</td>
</tr>
</tbody>
</table>

**Implementing custom edit codes for classic reports**

Users can implement custom edit codes that can be used when generating classic reports.

**About this task**

To implement custom edit codes:

**Procedure**

1. Create a new Java project and ensure that the Java build path contains the DataQuant API module located under the API installation location. For
example: productinstalldirectory\API\rocketsoftware. In order for this library to be available, the API component must be selected during the product installation.

2. For each custom edit code, create a class which implements the FormCustomEditCode interface.

3. Implement the following methods in this class:
   • Constructor must be without parameters or it can be omitted.
   • The init(int iColumnWidth, int iColumnAlignment, String strExtParams)
     method parameters are defined as follows:
     – iColumnWidth - the width in characters of a column with the edit code.
     – iColumnAlignment - the alignment of the column. The values that can be specified are ALIGN_LEFT, ALIGN_RIGHT, ALIGN_CENTER, ALIGN_DEFAULT.
     – strExtParams - typically it is a full string that was specified in the column's "Edit" parameter.
   • The register() method will return registration information for this custom edit code.
   • The format(Object obj) method formats the specified object as an array of strings. Every string in this array should be of iColumnWidth width and all alignment should be performed using space characters. You can use the StringUtils.getCharacterWidth(char, Locale) function to obtain a specified character's width.
   • destroy() - This method can be empty.

4. Generate a JAR file for this Java project and place it into the "user.home" directory. For example in Windows XP this directory would be: C:\Documents and Settings\Username\Application Data\IBM\DataQuant for Workstation.

5. Start DataQuant for Workstation. Select View > Preferences > Global Variables. From the Global Variable window specify a class name of the custom edit code for the DSQQW_RSUEDIT_JAR global variable. For example:
   com.ibm.custom.editcode.sample.VCAN1
   If you have several custom edit codes you must specify a class name for each separating each with a semi-colon. For example:
   com.ibm.custom.editcode.sample.VCAN1;com.ibm.custom.editcode.sample.VCAN2

6. Re-start the application.

Sample implementation for custom edit codes
The following sample class must be implemented to define each custom edit code. In the following sample VCAN1 is the name of the custom edit code.

The implemented class must have public constructor without parameters. Do not add any operations in constructor, use the init() method.

```java
package com.ibm.custom.editcode.sample;
import java.util.Date;
import java.util.Locale;
import com.ibm.bi.core.util.StringUtils;
import com.ibm.bi.qmf.core.model.form.FormCustomEditCode;
import com.ibm.bi.qmf.core.model.form.FormCustomEditCodeInfo;
```
public class VCAN1 implements FormCustomEditCode
{
    private int m_iWidth;
    private int m_iAlignment;
    private String m_strExtParams;

    /**This method is called after all operations to destroy structures*/
    public void destroy()
    {
    }

    /**This method is called to format data.
     * @param obj is an object to format. Parameter class can be String,
     * java.lang.Number, or java.util.Date depending on
     * data read from database or returned by NetRexx calculation.
     * You can restrict the passing of one or more of these types during
     * the registration of the custom edit code.
     * NaN, infinite (positive or negative), and null values can be passed
     * to this method if it was requested at registration of custom edit code.
     * The values are passed as strings in standard form. For example:
     * "DSQNULL" (for null values),
     * "DSQUNDEF" (for NaN values),
     * "DSQNOINS" (for LOB values).
     * @return returns formatted object as a string array. This array is a set of
     * text lines that will be output as a column value. Be sure that returned
     * lines have appropriate width. The width must be equal to iColumnWidth
     * parameter in the init() method.
     * @exception IllegalArgumentException (or any another RuntimeException) can
     * be thrown to show that the data type is incompatible with this edit code.
     * An exception can also be thrown for other reasons when the data
     * cannot be formatted by this edit code. In the case of an exception,
     * a "dummy string" (string of spaces) can be output as value
     * of this edit code.
     */
    public String[] format(Object obj) throws IllegalArgumentException
    {
        if(isNull(obj))
        {
            return formatNull();
        }
        else if(isNaN(obj))
        {
            return formatNaN();
        }
        else if(obj instanceof Number)
        {
            return format((Number)obj);
        }
        else if(obj instanceof Date)
        {
            return format((Date)obj);
        }
        else if(obj instanceof String)
        {
            return format((String)obj);
        }
        //Never raised
        throw new IllegalArgumentException();
    }

    private String[] format(Number num)
    {
        return align(num.toString());
    }
    private String[] format(String str)
    {
        return align(str);
    }
    private String[] format(Date date)
    {
        return align(date.toString());
    }
    private String[] formatNull()
    {
return align("<NULL>");
}
private String[] formatNaN()
{
    return align("<NAN>");
}
/**
 * This is a simple helper routine to determine if the object passed
 * to FormCustomEditCode.format() function represents null.
 */
public static Boolean isNull(Object obj)
{
    return "DSQNULL".equals(obj);
}
/**
 * This is a simple helper routine to determine if the object passed
 * to FormCustomEditCode.format() function represents NaN.
 */
public static Boolean isNaN(Object obj)
{
    return "DSQUNDEF".equals(obj);
}
/**
 * This is a simple helper routine to determine if the object passed
 * to FormCustomEditCode.format() function represents LOB.
 */
public static Boolean isLob(Object obj)
{
    return "DSQNOINS".equals(obj);
}
/**
 * This is a simple helper routine to determine if the object passed
 * to FormCustomEditCode.format() function represents special
 * objects like null, NaN or LOB.
 */
public static Boolean isSpecial(Object obj)
{
    return isNull(obj) || isNaN(obj) || isLob(obj);
}
/**You should implement this method to store the
 * column width and requested column alignment.
 * This method is called before
 * any format operation.
 */
public void init(int iColumnWidth, int iColumnAlignment, String strExtParams)
{
    m_iWidth = iColumnWidth;
    m_iAlignment = iColumnAlignment;
    m_strExtParams = strExtParams;
}
private String[] align(String strLine)
{
    return align(strLine, m_iAlignment, m_iWidth);
}
/**
 * Simple alignment routine.
 * This routine operates on a single line of text.
 */
public static String[] align(String strLine, int alignment, int iWidth)
{
    switch (alignment)
    {
    case ALIGN_CENTER:
    {
        return new String[] {alignStringToCenter(strLine, iWidth)};
    }
    case ALIGN_RIGHT:
    {
        return new String[] {alignStringToRight(strLine, iWidth)};
    }
    case FormCustomEditCode.ALIGN_LEFT:
case FormCustomEditCode.ALIGN_DEFAULT:
    default:
    {
        return new String[]{alignStringToLeft(strLine, iWidth)};
    }
}
/**
 * Formats string, adding ' ' character to the right
 * and to the left so the resulting length is iWidth
 */
private static String alignStringToCenter(String strSource, int iWidth)
{
    StringBuffer sbSource = new StringBuffer();
    int iLength = ensureStringToFit(strSource, iWidth, sbSource);
    int iAddLen = iWidth - iLength;
    if (iAddLen >= 0)
    {
        return fillSpaces(sbSource, iAddLen / 2, iAddLen - (iAddLen / 2));
    }
    return sbSource.toString();
}
/**
 * Formats string, adding ' ' character to the left so
 * the resulting length is iWidth
 */
private static String alignStringToRight(String strSource, int iWidth)
{
    StringBuffer sbSource = new StringBuffer();
    int iLength = ensureStringToFit(strSource, iWidth, sbSource);
    int iAddLen = iWidth - iLength;
    if (iAddLen >= 0)
    {
        return fillSpaces(sbSource, iAddLen, 0);
    }
    return sbSource.toString();
}
/**
 * Formats string, adding ' ' character to the right so
 * the resulting length is iWidth
 */
private static String alignStringToLeft(String strSource, int iWidth)
{
    StringBuffer sbSource = new StringBuffer();
    int iLength = ensureStringToFit(strSource, iWidth, sbSource);
    int iAddLen = iWidth - iLength;
    if (iAddLen >= 0)
    {
        return fillSpaces(sbSource, 0, iAddLen);
    }
    return sbSource.toString();
}
/**
 * Verify that a string fits into specified width
 */
private static int ensureStringToFit(String strSource, int iWidth, StringBuffer sbResult)
{
    Locale loc = Locale.getDefault();
    char[] carrSource = strSource.toCharArray();
    int iResultWidth = 0;
    for (int i = 0; i < carrSource.length; ++i)
    {
        char c = carrSource[i];
        //Calculate character width. 0 for BiDi managing characters,
        //2 for DBCS characters, 1 for others
        int iCharWidth = StringUtils.getCharacterWidth(c, loc);
        if (iResultWidth + iCharWidth > iWidth)
        {
            //Specified string does not fit into specified Width.
            //So, fill result with asterisks
            sbResult.setLength(0);
            sbResult.append(StringUtils.duplicateCharacter('*', iWidth));
        }
    }
}
private static String fillSpaces(StringBuffer sbResult, int iNumLeftSpaces, int iNumRightSpaces) {
    if (iNumLeftSpaces > 0) {
        sbResult.insert(0, StringUtils.duplicateCharacter(' ', iNumLeftSpaces));
    }
    if (iNumRightSpaces > 0) {
        sbResult.append(StringUtils.duplicateCharacter(' ', iNumRightSpaces));
    }
    return sbResult.toString();
}

/**
 * Append specified number of space characters to the right and to the left
 */
private static String fillSpaces(StringBuffer sbResult, int iNumLeftSpaces, int iNumRightSpaces) {
    if (iNumLeftSpaces > 0) {
        sbResult.insert(0, StringUtils.duplicateCharacter(' ', iNumLeftSpaces));
    }
    if (iNumRightSpaces > 0) {
        sbResult.append(StringUtils.duplicateCharacter(' ', iNumRightSpaces));
    }
    return sbResult.toString();
}

/**
* Implement this method to register the custom edit code.
* This method is called once for this class.
* (This method can be re-called if registration is
* requested again by the user application, for example changing registration
* file)
*/
public FormCustomEditCodeInfo register() {
    return new FormCustomEditCodeInfo("VCAN1", ",
        FormCustomEditCodeInfo.ACCEPT_SPECIAL_VALUES | FormCustomEditCodeInfo.ACCEPT_NUMBERS | FormCustomEditCodeInfo.ACCEPT_DATES | FormCustomEditCodeInfo.ACCEPT_STRINGS | FormCustomEditCodeInfo.AUTOCONVERT_TO_STRING);
}

Edit codes for metadata

You can use the metadata edit code M to display in a report the descriptive data for a column rather than the actual data.

<table>
<thead>
<tr>
<th>Edit Code</th>
<th>Format</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>M</td>
<td>Displays in character format a column's metadata.</td>
<td>The metadata for a column is found in the Descriptor Area (DA) and consists of the type and the length of the data that will be included in the column. If a column with the edit code M is null, a null indicator is displayed rather than the metadata. If the column size is less than the amount needed to display the metadata, the metadata is truncated in order to fit into the column space.</td>
</tr>
</tbody>
</table>
Chapter 18. Customizing the query environment

There are many ways that you can customize your query and reporting environment.

Using the command line

The command line interface allows DataQuant for Workstation users to connect to a repository and run objects by submitting commands using their operating command line rather than the application interface.

About this task

The following steps describe how to connect to a secured repository and run objects using the operating system command line:

Procedure

1. Create a procedure. For example:
   
   ```plain
   run query ObjectTracking (form=ObjectTrackingForm
   export form to c:\dqtrack.frm
   ```
   
   This example executes a query called ObjectTracking using the form called ObjectTrackingForm for form generation. The procedure exports the form to a file called dqtrack.frm in the c:\ directory.

2. Save the procedure in a secured repository with the name proc12, save the procedure in a QMF catalog with the name DB2INST1.Procedure1, and save as a local file with the name of Procedure1.prc.

3. There are several parameters that you can specify when you create your run command. The following examples use some of the parameters and illustrate how to run a procedure object that has been saved in a repository, a QMF catalog and a local file:
   
   - Type the following command in the operating system command line to connect to a secured repository when the login and password for the repository storage are saved in the repository connection named Connection1; the repository user ID repuserid; and the password reppassword and run a procedure named proc12 that resides in the repository against the data source DB2AIX using the user ID db2inst1 and password db2inst1 to access the data source:
     
     ```plaintext
     "C:\Program Files\IBM\IBM DataQuant\DataQuant for Workstation\dataquant.exe" /RConnection:Connection1 /RUser:repuserid /RPassword:reppassword /RObject:rsbi:/workspaces/Default/proc12 /IServer:DB2AIX /Run /IUserID:db2inst1 /IPassword:db2inst1
     ```
   
   - Type the following command in the operating system command line to connect to a secured repository when the login and password for the repository storage are not saved in the repository connection named Connection1 and the repository storage user ID rdbuserID and password rdbpassword and run a procedure named DB2INST1.Procedure1 that has been saved in a QMF catalog against the data source DB2AIX using the user ID db2inst1 and password db2inst1 to access the data source:
Type the following command in the operating system command line to connect to a repository with database-based security when the login and password for the repository storage are not saved in the repository connection named Connection1 and the repository storage user ID rdbuserID and password rdbpassword and run a procedure that contains a global variable named DB2INST1.Procedure1 that has been saved in a QMF catalog against the data source DB2AIX using the user ID db2inst1 and password db2inst1 to access the data source:

"C:\Program Files\IBM\IBM DataQuant\DataQuant for Workstation\dataquant.exe" /RConnection:Connection1 /RDBUser:rdbuserID /RDBPassword:rdbpassword /IObject:DB2INST1.Procedure1 /IServer:DB2AIX /Run /IUserID:db2inst1 /IPassword:db2inst1

Type the following command in the operating system command line to connect to a personal repository when the login and password for the repository storage are not required using the repository connection named Connection1 and run a procedure named Procedure1.prc that has been saved locally against the data source DB2AIX using the user ID db2inst1 and password db2inst1 to access the data source:

"C:\Program Files\IBM\IBM DataQuant\DataQuant for Workstation\dataquant.exe" /RConnection:Connection1 c:\Procedure1.prc /IServer:DB2AIX /Run /IUserID:db2inst1 /IPassword:db2inst1

List of command line parameters

This list includes each of the operating system command line parameters and a description of the parameter's function.

In the descriptions of each parameter, an 'object' refers to a query, form, procedure, visual dashboard, or visual report.

Table 29. Command line parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ObjectOwner.&quot;ObjectName&quot;</td>
<td>Use this parameter if the object that you will run resides in a QMF catalog.</td>
</tr>
<tr>
<td>/Object:&quot;ObjectOwner&quot;.&quot;ObjectName&quot;</td>
<td>Use this parameter if the object that you will run resides in a local or network file. You must specify the full path name.</td>
</tr>
<tr>
<td>/RObject:pathtoobject</td>
<td>Use this parameter if the object that you will run resides in a repository. You must specify the path to the object. For example: rsbi:/workspaces/default/query12 points to an object with the name query12 in a repository workspace called default. The path to any object in a repository is specified in the Key property found in the Properties window or view for the object. You open the Properties window by right-clicking an object in the Workspaces view.</td>
</tr>
<tr>
<td>/RConnection:connectionname</td>
<td>Name of the repository connection that will be used to connect to a repository.</td>
</tr>
<tr>
<td>/IServer:servername</td>
<td>The data source against which the object will be run. If this parameter is not specified, the last data source that was used will be accessed.</td>
</tr>
</tbody>
</table>
### Table 29. Command line parameters (continued)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>/Run</td>
<td>Start an opened or retrieved object.</td>
</tr>
<tr>
<td></td>
<td><strong>Note:</strong> If a form object does not have an associated data source, the /Run parameter is ignored.</td>
</tr>
<tr>
<td>/IDisplay</td>
<td>Display the object. If the /Run or /IDisplay parameters are not specified, then the /IDisplay parameter will be used.</td>
</tr>
<tr>
<td>/RUser:ruserID</td>
<td>The user ID that will be used to connect to any secured repository, except a repository with database-based security.</td>
</tr>
<tr>
<td>/RPassword:rpassword</td>
<td>The password that will be used with the user ID specified in the /RUser parameter.</td>
</tr>
<tr>
<td>/RDBUser:rdbuserID</td>
<td>The user ID that will be used to connect to the shared repository and authenticate user in the repository storage.</td>
</tr>
<tr>
<td>/RDBPassword:rdbpassword</td>
<td>The password that will be used with the user ID specified in the /RDBUser parameter.</td>
</tr>
<tr>
<td>/IUserID:userID</td>
<td>The user ID that will be used to access the data source against which the object will be run.</td>
</tr>
<tr>
<td>/IPassword:password</td>
<td>The password that will be used with the user ID specified in the /IUserID parameter.</td>
</tr>
<tr>
<td>/Batch</td>
<td>End the current session and close the application after running the object specified on the command line. <strong>Note:</strong> This parameter suppresses all warnings and error messages.</td>
</tr>
<tr>
<td>&quot;&amp;variablename=variablevalue&quot;</td>
<td>Define or update global variable values for a procedure or query. Multiple variables can be defined. <strong>Note:</strong> The variable name and value sent to the procedure must be enclosed in double quotes (&quot; &quot;), and preceded by an ampersand (&amp;).</td>
</tr>
<tr>
<td>/ICatalogUserID:cataloguserID</td>
<td>The user ID that will be used to access an extended QMF catalog.</td>
</tr>
<tr>
<td>/CatalogPassword:catalogpassword</td>
<td>The user password associated with the /ICatalogUserID parameter.</td>
</tr>
</tbody>
</table>

---

**Using the command bar**

You can run procedure commands directly from the DataQuant for Workstation interface using the command bar.

**About this task**

To run a procedure command from the command bar:

**Procedure**

1. Open the command bar by clicking the **Show Command Bar** toolbar button. If the command bar is closed, it will be opened. If the command bar is already open and visible, clicking the icon will close it. You can also toggle the command bar open and closed by selecting **View > Command Bar**.
2. Enter the command that you want to run in the Run command field of the command bar. You can type the command in the field or select from a drop-down list of previously used commands. The following procedure commands are available:
   - BOTTOM
   - CONNECT
   - CONVERT
   - DISPLAY
   - ERASE
   - EXPORT
   - FORWARD
   - IMPORT
   - PRINT
   - RESET GLOBAL
   - RUN
   - SAVE
   - SET GLOBAL
   - SHOW

3. Specify the name of a data source in the Data source field.

**Specifying default print options**

In DataQuant for Workstation, you must specify the page setup options that will be used when you print the contents of the available window.

**About this task**

If query results are in the window, then you will use the Query Results Page Setup dialog. If a form is in the available window, you will use the Form Page Setup dialog box.

To control the appearance of printed pages for query results and classic reports:

**Procedure**

1. Open the Page Setup dialog box by selecting File > Page Setup.
2. Specify the default printing options that will apply to your object.
3. Click Set as Default to save the setup values that you entered. The new values will become the default values.
4. Click OK to close the Page Setup dialog box. The options that you specified are used to control the appearance of the printed object.

**Viewing resource limits**

DataQuant resource limits control your data source access and resource consumption.

**About this task**

You can view the resource limits that are in effect for your user ID. You must be connected to the data source in order to view the resource limits that are in effect for your user ID when you access that data source.
To view resource limits for your user ID:

**Procedure**

1. Select View > Resource Limits. The [Data Source]: Limits window opens. The information in the Active Resource Limits section shows whether the limits displayed are data source limits or query limits, and to what user and resource limit group they are applied.

2. Select the Timeouts page to see the warning and cancellation limits for idle queries, idle connections, and server response time that have been set for your user ID.

3. Select the Limits page to see the warning and cancellation limits for the maximum number of rows and bytes of data that can be retrieved using your current user ID and the maximum number of simultaneous connections DataQuant can establish to the database server for your user ID.

4. Select the SQL Verbs page to see the SQL verbs that your user ID is allowed to use when a particular schedule is in effect.

5. Select the Options page to see what server access options have been set for your user ID. The server access options include: database access using the DataQuant user interface and programming interface; database update confirmation; isolation levels for user queries; mandatory account information; and the ability to fetch all rows automatically, export data, edit tables, and only run saved queries.

6. Select the Save Data page to see if you are allowed to save data at the database.

7. Select the Binding page to see if you are allowed to bind or drop static packages for your queries.

8. Select the Object Tracking page to see if DataQuant object tracking is activated for your user ID.

9. Select the LOB Options page to see if you can access and retrieve data from tables containing large object data (LOB).

10. Select the Cache page to control caching options for each user.

11. Select the Object list page to define the default owner name to be used to filter data source objects.

12. Click OK to close the Resource Limits for [user ID] window.

**About resource limits**

Resource limits govern your access to a data source based on the user ID that you use to access the data source.

Your user ID can belong to multiple resource limits groups. In addition, your user ID can belong to multiple resource groups that are defined for multiple data sources. Each resource limits group that you belong to can have one or more schedules, each with a different set of resource limits. The schedule in effect specifies what associated set of resource limits are in effect for your user ID when you access a data source.

Most resource limits are checked before a query leaves the workstation. The query never reaches your target database for evaluation if one of these resource limits is violated. Other resource limits are evaluated after your target database has started to run the query. In this case, any database limiting rules take precedence over DataQuant resource limits.
The resource limits for your user ID are set by your DataQuant administrator. Only the person who administers DataQuant can change any of the resource limits values. All of the information displayed is read-only.

**Specifying Timeouts resource limits**

The Timeouts resource limits set the timeout warning and cancellation limits for idle queries, idle connections, and data source response time.

**Idle Query Timeouts** and **Server Response Timeouts** resource limits can be set for both data sources and for individual queries. The **Idle Connection Timeouts** resource limit can only be set for data sources and not for individual queries. The Timeouts resource limits are as follows:

**Idle Query Timeouts**

These fields specify the amount of time that a query can remain idle. The Warning limit field displays the number of seconds that a query can remain idle before the user is informed and asked if the query should be canceled. The Cancel limit field displays the number of seconds that a query can remain idle before the query is automatically canceled.

**Server Response Timeouts**

These fields specify the amount of time that can be spent waiting for a response from the database. The Warning limit field specifies the number of seconds that can elapse before the user is informed and asked if the database request should be canceled. The Cancel limit field specifies the number of seconds that can elapse before the database request is automatically canceled.

*Note:* A lower timeout limit prevents long running, runaway queries. A higher timeout limit allows database requests to complete when the database server is slow due to resource contention or other reasons.

**Idle Connection Timeouts**

This field specifies the amount of time that an idle connection to a database data source is retained. The Cancel limit field specifies the number of seconds that an idle connection is retained before it is automatically closed.

*Note:* A lower timeout limit minimizes the resources consumed at the database by idle connections. A higher timeout limit minimizes the overhead of establishing connections.

**Specifying Limits resource limits**

The Limits resource limits set the warning and cancellation limits for the maximum number of rows and bytes of data that can be retrieved from the database data source; and the maximum number of simultaneous connections that can be established to the data source.

Limits resource limits can be set for data sources. The Limits resource limits are as follows:

**Maximum Rows to Fetch**

These fields specify the limits for the number of rows of data that can be retrieved from a database data source when running a query. The Warning limit field specifies the number of rows that can be retrieved from the data source before you are notified that a row limit has been reached. The user is prompted to select whether more data should be fetched or should the
query be canceled. The **Cancel limit** field displays the number of rows that can be retrieved from the data source before the row limit is reached. The query is then automatically cancelled.

**Maximum Bytes to Fetch**

These fields specify the limits for the number of bytes of data that can be retrieved from a database data source when running a query. The **Warning limit** specifies the number of bytes that can be retrieved from the data source before the user is notified that the byte limit has been reached. The user is then prompted to specify whether more data should be fetched or should the query be canceled. The **Cancel limit** displays the number of bytes that can be retrieved from the data source before the byte limit is reached. The query is then automatically cancelled.

**Maximum Connections**

This field specifies the limit for the number of simultaneous connections that can be established to a database data source. The **Cancel limit** specifies the number of simultaneous connections that can be established to a data source before the query requesting the next connection is automatically canceled. Setting the limit to zero (0) indicates that there is no limit to the number of connections.

Generally, connections to database data sources are reused. If you run one query at a data source, then run another query at the same data source, only one connection is required. However, if you run those two queries simultaneously, then two connections are required. If another connection to a data source is required and this limit is reached, an error is returned and the operation is not performed.

**Note:** To enable table editing, you must specify that two or more simultaneous connections are allowed.

**Specifying SQL Verbs resource limits**

The **SQL Verbs** resource limits specify the SQL verbs that you are allowed to use when the current schedule is in effect.

If a user tries to run a query that contains a *disallowed verb*, the query is canceled without sending the SQL to the data source. **SQL Verbs** resource limits can be set for data sources.

The use of the following SQL verbs can be controlled:

- ACQUIRE
- ALTER
- CALL
- COMMENT
- CREATE
- DELETE
- DROP
- EXPLAIN
- GRANT
- INSERT
- LABEL
- LOCK
- REFRESH
Specifying Options resource limits

The Options resource limits specify the levels of access you have to a data source and the data source objects.

Options resource limits can be set for data sources. The Options resource limits are as follows:

Allow access to data source from user interface
This field specifies whether you have permission to access a data source from the DataQuant for Workstation user interface when the current resource limits group schedule is in effect.

Allow access to data source from programming interface
This field specifies whether you have permission to access a data source from the DataQuant for Workstation application programming interface (API) when the current resource limits group schedule is in effect.

Fetch all rows automatically
This field specifies how the data source will send query results data to the application. By default, data is requested from the data source only as required to display query results. For example, if 20 rows fill up the query results window, only 20 rows are requested from the data source. When you scroll down to make the 21st row visible, more data is requested. If you run a query and then wait before scrolling down, the query remains active for that wait time, consuming resources at the data source. If this parameter is enabled, data is requested repeatedly until all data is received, independent of your scrolling requests.

Confirm database updates
This field specifies whether or not you must confirm all data source changes resulting from the queries that you run or the actions that you perform when editing tables. If this check box is selected, you will be prompted for confirmation before any data source changes can occur. If this check box is cleared, data source changes can occur without confirmation.

You can override this resource limit setting using the Confirm parameter in a Procedure command. The following table shows the correlation of the parameters, depending on the specified values.

<table>
<thead>
<tr>
<th>Confirm database updates value</th>
<th>Procedure parameter value</th>
<th>The confirmation dialog behavior</th>
</tr>
</thead>
<tbody>
<tr>
<td>Selected</td>
<td>confirm=yes</td>
<td>appears</td>
</tr>
<tr>
<td>Selected</td>
<td>confirm=no</td>
<td>does not appear</td>
</tr>
<tr>
<td>Selected</td>
<td>not specified</td>
<td>appears</td>
</tr>
<tr>
<td>Cleared</td>
<td>confirm=yes</td>
<td>appears</td>
</tr>
<tr>
<td>Cleared</td>
<td>confirm=no</td>
<td>does not appear</td>
</tr>
<tr>
<td>Cleared</td>
<td>not specified</td>
<td>does not appear</td>
</tr>
</tbody>
</table>
Enable exporting of data
This field specifies whether a user can export query result data to a file using the Export command from the File menu or the EXPORT DATA procedure command.

Enable table editing
This field specifies whether or not you can edit a table using the table editor feature.

Allow running saved queries only
This field specifies whether or not you can create and save new queries at the database data source. If checked, you can only run queries that are saved at the database data source. If clear, you can run queries, as well as create and save new queries at the database data source.

Isolation level for queries
The Isolation level for queries parameter only applies to databases that support transactions.

This field specifies the level of transaction isolation to be applied to a user's queries. There are varying levels of isolation, each with a differing degree of impact on transaction processing in the database.

Consult with your database administrator or the database server documentation to determine if the database server being used is configured to support transaction processing.

Valid values include:
• Default specifies that the isolation level for a query will be the default specified for all applications.
• Repeatable Read specifies that the isolation level for a query will be Repeatable Read (RR). The RR isolation level provides the most protection from other applications.
• All (RS) specifies that the isolation level for a query will be Read Stability (RS).
• Cursor Stability (CS) specifies that the isolation level for your query will be Cursor Stability (CS). The CS isolation level provides the maximum concurrency with data integrity.
• Change (UR) specifies that the isolation level for your query will be Uncommitted Read (RS) which allows an application to read uncommitted data.
• No Commit (NC) specifies that the isolation level for your query will be No Commit (NC). An application running with an isolation level of NC can not make permanent database changes.

Account
This field specifies the default account information that will be sent to a data source when you connect to that data source.

Account can be overridden
This field specifies whether you are allowed to override the default accounting information specified in the Account field when connecting to a data source. If selected, you can enter a new accounting string using the Set User Information window. If clear, you cannot enter new accounting information and you are prevented in some cases from accessing data source objects that are not owned by your resource limits group.
Require account information
This field specifies whether you must specify a valid accounting string in the Set User Information window before connecting to a database when this schedule is in effect.

Allow printing
This check box must be selected in order for you to print objects that are stored on the data source.

Specifying Save Data resource limits
The Save Data resource limits specify whether you as a member of the current resource limits group and using the current schedule can save data at the database data source.

Save Data resource limits can be set for data sources. The Save Data resource limits are as follows:

Allow save data command
This field specifies whether you can save data at the database data source. Saving data can be extremely resource-intensive. As a result it can have a significant impact on your database and network.

Default table space
This field specifies the default table space name that will be used for tables created with the save data command. This field is only available if the Allow save data command has been selected. The syntax of the table space name that you specify must conform to the database rules for table space names. Any value that you specify is used as part of the CREATE TABLE SQL statement that runs when you save data to a new table. Entering an asterisk (*) specifies that the default table space will be taken from a user's profile in the RDBI.PROFILE_VIEW.

Default table space can be overridden
This field specifies whether you can override the default table space. If selected, you can specify any table space name subject to database security authorizations. If this check box is not selected, you must use the table space name specified in the Default table space field.

Allow to replace any existing data
This field specifies whether a user can override the existing data when the objects are exported or imported to the database. If this option is selected, a user can replace the existing data with the new one in the database. If this check box is clear, a user can only append new data to the existing one.

Specifying Binding resource limits
The Binding resource limits specify whether you as a member of this resource limits group and using this schedule can bind or drop static packages for your queries to or from the data source.

Binding resource limits can only be set for data sources. The Binding resource limits are as follows:

Allow binding of packages
This field specifies whether or not you can bind static packages for your queries on the database data source.
Allow dropping of packages
This field specifies whether or not you can drop static packages from the database data source.

Default collection ID
This field specifies the default collection ID for the static packages that you bind.

Default collection ID can be overridden
This field specifies whether you must use the default collection ID. If this check box is selected, you can specify any collection ID subject to database security authorizations. If this check box is clear, you must use the default collection ID that is specified in the Default collection ID field.

Default isolation level
This field specifies the default isolation level for the static packages that you bind. You can select one of the isolation levels:
- Repeatable Read specifies that the default isolation level for the static packages will be Repeatable Read (RR). The RR isolation level isolates the SQL statements in the static package from the actions of concurrent users for rows the requester reads and changes, including phantom rows. RR isolation provides the most protection.
- All (RS) specifies that the default isolation level for the static packages will be Read Stability (RS). The RS isolation level isolates the SQL statements in the static package from the actions of concurrent users for rows the requester reads and changes.
- Cursor Stability (CS) specifies that the default isolation level for the static packages will be Cursor Stability (CS). The CS isolation level isolates the SQL statements in the static package and the current row to which the database cursor is positioned from the actions of concurrent users for changes the requester makes.
- Change (UR) specifies that the default isolation level for the static packages will be Uncommitted Read (UR). The UR isolation level isolates the SQL statements in the static package from the actions of concurrent users for changes the requester makes.
- No Commit (NC) specifies that the default isolation level for the static packages will be No Commit (NC). The NC isolation level does not isolate the SQL statements in the static package from the actions of concurrent users for changes the requester makes.

Default isolation level can be overridden
This field specifies whether you can override the isolation level specified in the Default isolation level field. If this check box is selected, you can specify any isolation level when binding packages. If this check is not selected, you must use the default isolation level specified in the Default isolation level field to bind static packages.

Specifying Object Tracking resource limits
The Object Tracking resource limits specify whether object tracking is activated for your user ID.

Object tracking must be enabled in order to collect the tracking data that is used to populate object tracking reports. Object tracking reports can help you locate unused objects, locate frequently accessed data sources such as tables or columns and spot potential problem areas. Object Tracking resource limits can be set for data sources.
The Object Tracking resource limits are as follows:

Enable summary object tracking
This field specifies whether summary object tracking is activated for your resource limits group. Summary tracking records the number of times an object is run, and the most recent times it was run and modified.

Enable detailed object tracking
This field specifies whether detailed object tracking is activated for your resource limits group. Detailed tracking records each time the object is run, who ran it, and the results. A large amount of data can be collected if this option is selected.

Note: This option must be enabled to use ad hoc object tracking or SQL text tracking.

Enable ad-hoc object tracking
This field specifies whether a record of each ad-hoc query is kept in the summary or detail object tracking tables. An ad-hoc query is a query that has not been named or saved at the database data source.

Note: The Enable detailed object tracking option must be enabled in order to enable this option.

Enable SQL text tracking
This field specifies whether a record of the SQL text of each query is kept in the detail tracking table.

Note: The Enable detailed object tracking option must be enabled in order to enable this option.

Last Used timestamp option
This field specifies when and by which commands data objects are updated.

Note: In cases when the object is saved to the QMF Catalog, the data is kept in the column LAST_USED of the table Q.OBJECT_DIRECTORY. If the object is saved to the repository, then the data is kept in the column LAST_USED of the RSBI.OBJECTS table.

The following options are available:

• Update when object is accessed (legacy mode)
  Data is updated when you run any of the following commands against the object:
  – CONVERT
  – DISPLAY
  – EXPORT
  – IMPORT
  – PRINT
  – RUN
  – SAVE

• Update when object is run, saved or imported
  Data is updated after performing RUN, SAVE or IMPORT commands.

• Update only when object is run
  Data is updated only after performing RUN command.
Last Used timestamp options can be overridden
Indicates whether the DSQEC_LAST_RUN variable value is ignored.
This check box is cleared by default.
Select this check box to change the variable value and override the
resource limit.

Specifying LOB Options resource limits
The LOB Options resource limits control a user's ability to access, retrieve, and
save data to and from tables containing large objects (LOBs).

A Large Object (LOB) is a DB2 for z/OS and DB2 for Linux, UNIX, and Windows
data type that houses non-traditional data such as text, multimedia, image, video,
photograph, sound, or any very large data file inside a database table. Retrieving
or saving LOB data can consume a substantial amount of resources. Even just
accessing a table that contains LOB data can be resource consumptive. Using the
LOB Options resource limits, you can control access to tables on the database data
source that include LOB data. LOB Options resource limits can be applied to both
data sources and individual queries.

The LOB Options resource limits are as follows:

LOB Retrieval Option
This field specifies whether a user can retrieve large object (LOB) data
when this schedule is in effect. You can select one of the following:

• Disable LOB columns specifies that users can not retrieve LOB data and
can not query or access any table that contains LOB data.

• Disable LOB data retrieval specifies that users can not retrieve LOB
data but can query tables that contain LOB data. Query result data will
be returned for all columns in the table except those that contain LOB
data.

• Retrieve LOB data on demand specifies that users can query tables that
contain LOB data and result data will be returned for all columns
including those columns that contain LOB objects. However, the results
data for the LOB columns will not actually be displayed in the column.
A pointer is displayed. When the user selects the pointer, the LOB data
will be displayed in the query results.
You can also pull the data from the database by double-clicking on the
pointer.

• Retrieve LOB data automatically specifies that users can query tables
that contain LOB data and the result data will be returned for all
columns, including those columns that contain LOB objects.
This option pulls ALL of the LOB data for ALL of the LOB columns
from the database to the local computer.
This option can potentially consume a large amount of resources and
time. The actual LOB data is not displayed in grids and reports. Instead,
pointers to the LOB data are displayed.

Enable saving LOB data
This field specifies whether users can save LOB data on the database data
source.

Maximum LOB column size
This field specifies the maximum size of a LOB column, in kilobytes, up to
2G (the maximum LOB size). The default is 0, no maximum. If a user queries a table with LOB data that is larger than the maximum, the LOB data will not be returned for display.

**LOB options can be overridden**

This field specifies whether a user can override the default LOB Options resource limits that have been specified for their group.

**Specifying Report Center resource limits**

You can view the level of QMF Report Center functionality allowed for members of a resource limits group on the **Report Center** page of the [Data Source]: Limits window.

With QMF Report Center you can generate custom reports using shared queries, forms, procedures and tables that are stored on the database server. The custom reports can then be viewed and manipulated in a variety of applications.

The **Report Center** resource limits are as follows:

**Allow display of all object types**

This option indicates whether members of this resource limits group are allowed to access the objects used for their QMF Report Center reports.

Users do not need special permission to use the QMF Report Center application or run reports. However, in order to create new reports they need access to the stored objects. Some users might not have permission to access these objects. If this option is set to **yes**, members of this resource limits group are allowed to access the objects that are stored on the database server and create new reports.

**Note:** The **Allow display of all object types** option is set to **yes** by default.

**Allow administration of root node**

This option indicates whether or not the members of this resource limits group are authorized to add folders to the root folder, **Public Favorites**, used by the QMF Report Center application. The Public Favorites folder is stored on the database server.

QMF Report Center uses two top-level folders, **Personal Favorites** and **Public Favorites**, to store user's reports. Each folder contains reports that point to objects that reside on the database server. The objects are not contained within the folders.

The **Personal Favorites** folder resides locally on the user's machine. This folder and the reports contained in it can only be accessed by the user who created it.

The **Public Favorites** folder resides on the database server. It can be accessed (potentially) by all QMF Report Center users. However, in order to add a first-level sub folder to the **Public Favorites** folder, users must have permission. This option gives the members of this resource limits group that permission.

If granted permission to create a first-level sub folder, the member of the resource limit group is authorized to use the QMF Report Center security options to control who can run or amend the reports residing in the folder. The **Public Favorites** folder can have up to 50 first-level sub folders.
Note: The **Allow administration of root node** option is set to *no* by default.

**Allow taking ownership of node**
This option indicates whether or not all members of this resource limits group have authorization to take ownership of a folder or a report in the *Public Favorites* folder.

If this option is set to *yes*, users can access any report or folder in the *Public Favorites* folder regardless of whether they have been granted that access by QMF Report Center security.

Note: The **Allow taking ownership of node** option is set to *no* by default.

### Specifying Cache resource limits
The **Cache** resource limits control caching for query results.

With this resource limit enabled, when a query is run, the results will be cached. If the query is run again, and nothing has changed, the results will be taken from the cached results rather than run again at the data source. **Cache** resource limits can be applied to both data sources and individual queries.

The **Cache** resource limits are as follows:

**Cache is enabled**
This field specifies whether the caching of result set data is enabled. Caching is disabled by default. If caching is enabled, result data from a query is cached (stored) and is available for use by other users that have the same privileges to access this data on the data source. Caching result set data can significantly decrease resource consumption and improve performance. The cached result set data is not automatically refreshed.

**Auto data fetching enabled**
This field specifies how the cached rows of data will be retrieved. This resource limit becomes available if **Cache is enabled** is selected. When selected, this option specifies that if the query result set is taken from the cache, the entire result set will be presented. When this option is not selected, 100 rows (the default) of the query result set is presented.

**Data expiration interval**
This field specifies the time period the query result set data will remain in cache. You can set the time period by specifying the number of days, hours, and minutes. After this period, the cached query result set data will be updated.

**Cache options can be overridden**
This field specifies whether the cache options can be overridden.

### Specifying Object List resource limits
The **Object List** resource limits specifies default and allowed owner names for data sources.

With **Object List** resource limits, you can define the default owner name that will be used to filter data source objects. You can also specify a specific list of owner names that can be used to filter data source objects aside from the default name.

The **Object List** resource limits are as follows:
Default Owner Name
This field specifies the default owner name that will be used to filter the data source object list.

Default Owner Name can be overridden
This check box allows the user to override the current default owner name.

Allowed Owner Names
This field specifies the list of additional owner names that can be used to filter the data source object list. Owner names should be separated by a comma.

Allow <USERNAME>
This check box allows the current user’s owner name to be used to filter the data source object list.

Allow <SQLID>
This check box allows the current user’s SQLID to be used to filter the data source object list.
Appendix A. Accessibility

Software accessibility features help those with physical disabilities, such as restricted mobility or limited vision, use their computers.

Note: The DataQuant for Workstation application provides the best accessibility functionality. Users with physical disabilities who must make use of the DataQuant accessibility features should install and run DataQuant for Workstation, as opposed to DataQuant for WebSphere.

Standard keyboard equivalents in DataQuant for Workstation

Keyboard equivalents use keyboard keys to perform mouse actions instead of using a mouse.

The keyboard is the most frequently used alternative for performing mouse functions. DataQuant supports the standard keyboard equivalents that include:

• Shortcut, or accelerator keys, to perform the most frequently used functions in pull-down menus without going to the menu. For example:
  – Ctrl+S to save
  – Ctrl+P to print
  – Ctrl+R to run a query
  Shortcuts display in the pull-down menu next to its function.

• Mnemonics, or access keys, are available to perform each function on a menu or window. A mnemonic for a function is the underlined character in the function name. For example:
  – Press F to open the File menu
  – Press O to open the Open window
  Use ALT to activate the mnemonic and move the keyboard focus.

For more information, refer to the documentation for your operating system for a complete list of standard keyboard equivalents.

Standard keyboard navigation

Keyboard navigation refers to using keys instead of a mouse to move from item to item on your screen.

The movement is usually in the order specified by the operating system or your application. DataQuant applications follow the standards with regard to the typical keys used for keyboard navigation such as using TAB and SHIFT+TAB to move between controls, and arrow keys to move up, down, and sideways between items.

Operating system accessibility

Most operating systems have a set of accessibility options that enable individuals with disabilities to customize system-wide settings.

DataQuant applications:
• Inherit their settings from the operating system
- Do not interfere with keyboard accessibility features built into the operating system

**Assistive technology products**

DataQuant applications support assistive technology products, such as screen readers and voice synthesizers.

Each DataQuant application interface requires special navigation when using a screen reader with query results. See “Navigation in DataQuant for Workstation” or “Navigation in DataQuant for WebSphere” on page 470 for more information.

### Navigation in DataQuant for Workstation

The following table shows how to accomplish various tasks using keystrokes.

<table>
<thead>
<tr>
<th>To:</th>
<th>Process:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Activate mnemonics in menus</td>
<td>Press the <strong>Alt</strong> key. Mnemonics in each menu are activated.</td>
</tr>
<tr>
<td>Obtain online help</td>
<td>Press <strong>F1</strong> to open a specific help task or <strong>Alt+H</strong> to open the <strong>Help</strong> menu. Use the arrow keys to move through the menu selections. Press <strong>Enter</strong> to open a menu command.</td>
</tr>
<tr>
<td>Display keyboard shortcuts</td>
<td>Press <strong>Ctrl+Shift+L</strong>.</td>
</tr>
<tr>
<td>Display the Draw Query wizard</td>
<td>Press <strong>Ctrl+D</strong>.</td>
</tr>
<tr>
<td>Display the switch list of the editors of the opened DataQuant objects and switch between the editors</td>
<td>Press <strong>Ctrl+F6</strong> to move forward in the switch list and <strong>Ctrl+Shift+F6</strong> to move in the opposite direction.</td>
</tr>
<tr>
<td>Left to right switch between the tabs of the opened DataQuant objects</td>
<td>Press <strong>Ctrl+Page Up</strong></td>
</tr>
<tr>
<td>Right to left switch between the tabs of the opened DataQuant objects</td>
<td>Press <strong>Ctrl+Page Down</strong></td>
</tr>
<tr>
<td>Move focus through each element</td>
<td>Press <strong>TAB</strong> to move forward or <strong>SHIFT+TAB</strong> to move in the opposite direction.</td>
</tr>
<tr>
<td>Emulate clicks on a link</td>
<td>Use the <strong>TAB</strong> key to jump from link to link, then press the <strong>Enter</strong> key to open the link.</td>
</tr>
<tr>
<td>Emulate clicks on a button</td>
<td>Use the <strong>TAB</strong> key to jump from button to button, then press <strong>Space</strong> key to activate the button.</td>
</tr>
<tr>
<td>Within a window, activate the default action</td>
<td>Press the <strong>Enter</strong> key.</td>
</tr>
<tr>
<td>Within a window, cancel the action</td>
<td>Press the <strong>ESC</strong> key.</td>
</tr>
<tr>
<td>Left to right switch between views</td>
<td>Press <strong>Ctrl+F7</strong></td>
</tr>
<tr>
<td>To:</td>
<td>Process:</td>
</tr>
<tr>
<td>--------------------------------------------------------------------</td>
<td>--------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Right to left switch between views</td>
<td>Press Ctrl+Shift+F7</td>
</tr>
<tr>
<td>Left to right switch between perspectives</td>
<td>Press Ctrl+F8</td>
</tr>
<tr>
<td>Right to left switch between perspectives</td>
<td>Press Ctrl+Shift+F8</td>
</tr>
<tr>
<td>Left to right switch between tabs in the active editor</td>
<td>Press Alt+Page Up</td>
</tr>
<tr>
<td>Right to left switch between tabs in the active editor</td>
<td>Press Alt+Page Down</td>
</tr>
<tr>
<td>Activate command bar</td>
<td>Press Ctrl+Shift+C</td>
</tr>
<tr>
<td>Use screen reader for query results or Use browser to display</td>
<td>1. Run the query.</td>
</tr>
<tr>
<td>query results in high contrast mode for the visually impaired</td>
<td>2. Press Alt+R to open the Results menu</td>
</tr>
<tr>
<td></td>
<td>3. Select Export to open the Export Query Results wizard.</td>
</tr>
<tr>
<td></td>
<td>4. Use the TAB key to move through controls. Use arrow keys</td>
</tr>
<tr>
<td></td>
<td>to move through selection items in the control. Select HTML</td>
</tr>
<tr>
<td></td>
<td>from the Export Type list. Specify where to save the file in the</td>
</tr>
<tr>
<td></td>
<td>File name field.</td>
</tr>
<tr>
<td></td>
<td>5. Press Finish.</td>
</tr>
<tr>
<td></td>
<td>6. Open the *.htm file in a browser.</td>
</tr>
<tr>
<td></td>
<td>or</td>
</tr>
<tr>
<td></td>
<td>1. Run the query.</td>
</tr>
<tr>
<td></td>
<td>2. Press Alt+R to open the Results menu</td>
</tr>
<tr>
<td></td>
<td>3. Select Export to open the Export Query Results wizard.</td>
</tr>
<tr>
<td></td>
<td>4. Use the TAB key to move through controls. Use arrow keys</td>
</tr>
<tr>
<td></td>
<td>to move through selection items in the control. Select CSV</td>
</tr>
<tr>
<td></td>
<td>from the Export Type list. Specify where to save the file in the</td>
</tr>
<tr>
<td></td>
<td>File name field.</td>
</tr>
<tr>
<td></td>
<td>5. Press Finish.</td>
</tr>
<tr>
<td></td>
<td>6. Open the *.csv file in supportive software such as Microsoft Excel.</td>
</tr>
<tr>
<td>Add object or condition</td>
<td>Use TAB to switch between different sections of the Query Builder editor,</td>
</tr>
<tr>
<td></td>
<td>and then press Spacebar to expand the sections.</td>
</tr>
<tr>
<td></td>
<td>2. Press Shift+Arrow to select additional, consecutive rows.</td>
</tr>
<tr>
<td></td>
<td>3. Press Shift+Enter to add the objects.</td>
</tr>
<tr>
<td>Select multiple objects</td>
<td>1. Tab to the list box.</td>
</tr>
<tr>
<td></td>
<td>2. Press Shift+Arrow to select additional, consecutive rows.</td>
</tr>
<tr>
<td></td>
<td>3. Press Shift+Enter to add the objects.</td>
</tr>
<tr>
<td>Edit rows in list boxes (only for rows that can be edited)</td>
<td>1. TAB to the list box, such as the Global Variables window.</td>
</tr>
<tr>
<td></td>
<td>2. Press the spacebar or use the up and down arrow keys to select the</td>
</tr>
<tr>
<td></td>
<td>row.</td>
</tr>
<tr>
<td></td>
<td>3. Press F2 to activate edit mode.</td>
</tr>
<tr>
<td></td>
<td>4. Use TAB to move between columns and rows.</td>
</tr>
<tr>
<td></td>
<td>5. Press Enter to accept edits.</td>
</tr>
<tr>
<td></td>
<td>6. Press TAB to exit the list box and move to the next control in the</td>
</tr>
<tr>
<td></td>
<td>dialog.</td>
</tr>
</tbody>
</table>

Appendix A. Accessibility  469
### Navigation in DataQuant for WebSphere

The following table shows how to accomplish various tasks using keystrokes.

<table>
<thead>
<tr>
<th>To:</th>
<th>Process:</th>
</tr>
</thead>
</table>
| **Edit query results**                   | 1. Press **CTRL+F7** or **CTRL+SHIFT+F7** until either the **Workspaces** or **Repository Explorer** view is highlighted.  
2. Use the down or up arrow keys to select the folder that contains a table.  
3. Use the right arrow key to expand folders and the left arrow key to collapse folders.  
4. Use the arrow keys to expose and select the table that you want to edit.  
5. Press the **Menu** key (the key to the left of the right-side of the **CTRL** key).  
6. Press the mnemonic key for Open With (the **h** key).  
7. Use the down arrow key to highlight **Table Editor**.  
8. Press the **ENTER** key. The table opens in **Table Editor**. |
| **Change font for query results**        | 1. Run the query.  
2. Press **Alt+A** to select all.  
3. Press **Alt+R** to open the **Results** menu.  
4. Press **F** to open the Font window. |

---

### To: Process:

<table>
<thead>
<tr>
<th>Move focus through each element</th>
<th>Press <strong>TAB</strong> to move forward or <strong>SHIFT+TAB</strong> to move in the opposite direction.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Emulate clicks on a link</td>
<td>Use the <strong>TAB</strong> key to jump from link to link, then press the <strong>Enter</strong> key to open the link.</td>
</tr>
<tr>
<td>Emulate clicks on a button</td>
<td>Use the <strong>TAB</strong> key to jump from button to button, then press <strong>Space</strong> key to activate the button.</td>
</tr>
<tr>
<td>Within a dialog, activate the default action</td>
<td>Press the <strong>Enter</strong> key.</td>
</tr>
<tr>
<td>Within a dialog, cancel the action</td>
<td>Press the <strong>ESC</strong> key.</td>
</tr>
<tr>
<td>Move focus to the command line</td>
<td>Press <strong>ALT+C</strong>.</td>
</tr>
<tr>
<td>Move focus to the main menu</td>
<td>Press the left <strong>ALT</strong> key. Navigate through the menu commands using the arrow keys.</td>
</tr>
<tr>
<td>Open pop-up menus</td>
<td>Press the <strong>Menu</strong> key if you have an extended keyboard. Press the right <strong>Ctrl</strong> key. Navigate through the menu commands using the arrow keys.</td>
</tr>
<tr>
<td>Activate top-level menu</td>
<td>Press <strong>Shift+Ctrl+1 (Shift+Ctrl+2, Shift+Ctrl+3, and so on)</strong></td>
</tr>
<tr>
<td>Switch between open editors</td>
<td>Press <strong>Shift+Ctrl+F6</strong></td>
</tr>
<tr>
<td>Switch between open views</td>
<td>Press <strong>Shift+Ctrl+F7</strong></td>
</tr>
</tbody>
</table>
Appendix B. Troubleshooting

Use these topics to diagnose and correct problems that you experience with DataQuant.

Problem displaying images in DataQuant for WebSphere using Internet Explorer 8

Symptoms
Visual report and visual dashboard images do not display properly in DataQuant for WebSphere when using Internet Explorer version 8.

Environment
DataQuant for WebSphere

Resolving the problem
Delete files from the Temporary Internet files folder in the Internet Explorer browser:

1. Select Tools > Delete Browsing History from the menu bar. The Delete Browsing History dialog opens.
2. Check the Temporary Internet files check box.
3. Click Delete. The Delete Browsing History dialog closes and the files are deleted.
4. Reopen the visual report or visual dashboard.

Problem displaying list of objects after upgrading to DB2 UDB for z/OS V8 that supports long names

Symptoms
Prior to DB2 for z/OS V8, object names had to be 8 characters long CHAR(8). If an object name was less than 8 characters long, DB2 for z/OS made up the difference by appending spaces to the end of the name. For example, if an object name was 5 characters long, DB2 for z/OS added 3 spaces to the end of the name. DB2 for z/OS V8 added support for long names. In DB2 for z/OS V8, object names are defined as VARCHAR(128). Object names that were appended with spaces may still exist in your database. To ensure that object names appended with spaces are searchable, you need to run UPDATE statements on the objects. If you do not run the UPDATE statements, object names that were appended will not display in query results.

Environment
DataQuant for WebSphere and DataQuant for Workstation

Resolving the problem
1. Before implementing the fix, back up the following tables:
   • Q.OBJECT_DIRECTORY
   • Q.OBJECT_REMARKS
   • Q.OBJECT_DATA
2. Run the following UPDATE statements:
Creating a response file

To run a silent installation of DataQuant for Workstation, you must create a response file that specifies all installation options. This file must specify all the features that you want to install and installation paths.

About this task

To create a response file:

Procedure

1. In the command line, run the installation command.
   - See the example for a Windows installation to the current directory:
     `setupwin32.exe -r installer.properties`
     where: `-r` specifies that the installation software runs in record mode.
   - See the example for a Windows installation to the custom directory:
     `setupwin32.exe -r C:\drive_path\response_file.properties`
     where:
     - `-r` specifies that the installation software runs in record mode.
     - `C:\drive_path\response_file.properties` specifies the path to the response file that you want to create.

2. As the installation proceeds, you select the features that each user that runs this executable installed on their machine.

3. Upon completion of the installation, the application is installed in the directory that you specified during the installation.

Results

The response file is created and saved in the current directory or the directory that you specified in the path.

What to do next

Distribute the files to the user machines to that you want to install the software and run the silent installation.

DataQuant for Workstation Win32 API reverts to previous version of QMF for Windows

Symptoms

If an earlier version of QMF for Windows that is installed alongside DataQuant for Workstation Version 1 is opened, API instances automatically revert to QMF for Windows, even after Version 1 has been reopened.

Causes

When DataQuant for Workstation Version 1 is installed, the application version number is updated in the system registry. When a preexisting version of QMF for
Windows is opened, it automatically reverts the application’s version number in the system registry. When Version 1 is opened, this automatic rename does not take place.

**Resolving the problem**

Any time that the QMF for Windows application is opened, you must manually reset the version number in the system registry if you want API instances to point to DataQuant for Workstation Version 1.

**User response:** To reset the application version number in the system registry:
1. Go to the following location in your workstation directory:
   
   "C:\Program Files\IBM\IBM DataQuant\API"

2. Double-click the file named qmfwin.exe

**Note:** Win32 API uses active repository connections. If there is not an active repository connection, the system opens up a dialog box that lists all available connections where you must select the connection you want to use.
Appendix C. Messages

This topic lists and describes the messages and return codes issued by the program components of DataQuant for Workstation and DataQuant for WebSphere.

DataQuant for Workstation and DataQuant for WebSphere messages are numbered as follows:

*CPT*prefix*MESS*number

**CPT**<br>
The abbreviation of the component of the application that threw the error. This can be an indication of the first place to look in order to resolve the error.

**MESS**<br>
The specific number of the error message.

For example: ABC0001

All of the error messages of each component are listed in alphabetical order. To look up more information about a specific error message and how to resolve it, search for the component prefix and number of the message in this publication.

**Note:** Since DataQuant supports DB2 databases, DB2 error messages are included in the application. DB2 error messages have a component prefix of DB2. (for example, DB20001) If you encounter a message with this component prefix, refer to the DB2 online documentation at [www.ibm.com](http://www.ibm.com). If you are using the DB2 Type 4 UDB driver or the DB2 Universal JDBC Driver, to receive extended SQL exception error messages, check the retrieveMessagesFromServerOnGetMessage option in the Advanced JDBC Settings window of the Edit Repository Connection wizard.

<table>
<thead>
<tr>
<th>API1001</th>
<th>{0}</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>This is a container for other messages.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>No action is required.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>API1002</th>
<th>The API has not been initialized.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The API instance has not been initialized. This could mean that the QMFwin API was not initialized, or a repository connection was not established.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Check the QMFwin API logs to determine the error that caused the API instance to fail, or establish a repository connection.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>API1003</th>
<th>No data found.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>There was an attempt to obtain a row that does not exist in a result set.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>No action required.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>API1006</th>
<th>The specified source {0} is not valid.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An invalid value was specified for the SourceType parameter of the initializeProc() or initializeQuery() function.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Specify a valid value for the SourceType parameter of any initializeProc() or initializeQuery() function.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>API1007</th>
<th>The specified source name is not valid.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An invalid value was specified for the Source, CollectionName, PackageName, or ConsistencyToken parameter of an initializeProc(), initializeQuery(), or initializeStaticQuery() function.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Specify a valid value for the Source, CollectionName, PackageName, or ConsistencyToken parameter of any initializeProc(), initializeQuery(), or initializeStaticQuery() function.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>API1008</th>
<th>The specified database server {0} is not valid.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An invalid value was specified for the Source, CollectionName, PackageName, or ConsistencyToken parameter of an initializeProc(), initializeQuery(), or initializeStaticQuery() function.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Specify a valid value for the Source, CollectionName, PackageName, or ConsistencyToken parameter of any initializeProc(), initializeQuery(), or initializeStaticQuery() function.</td>
</tr>
</tbody>
</table>
Explanation: An unknown data source alias was specified in a getServerAttributes(), getServerType() or initializeServer() function.

User response: Specify a valid data source alias for any getServerAttributes(), getServerType() or initializeServer() function.

API1009  No database server has been specified.
Explanation: The initializeServer() function was not invoked before other functions were invoked that require an initialized data source.

User response: Ensure that the initializeServer() function is invoked before other data source dependent functions are called.

API1010  The specified row limit (0) is not valid.
Explanation: An invalid value was specified for the RowLimit parameter of an open() function.

User response: Specify a positive value for the RowLimit parameter of any open() function.

API1011  The specified query ID (0) is not valid.
Explanation: An invalid value was specified for the QueryID parameter of an API function.

User response: Specify a valid value for the QueryID parameter of all relevant functions.

API1012  The SQL verb (0) cannot be used with open().
Explanation: An open() function was called for a query that does not contain the SELECT, VALUES, WITH, or CALL SQL verbs.

User response: Ensure that any open() function calls a query that contains one of the SQL verbs listed in the error message text.

API1013  The SQL verb (0) cannot be used with execute().
Explanation: An execute() function was called for a query that contains the SELECT, VALUES, WITH, or CALL SQL verbs.

User response: Ensure that any execute() function does not call a query that contains one of the SQL verbs listed in the error message text.

API1015  A new procedure could not be created. There may be too many documents already open, or there may be insufficient memory.
Explanation: An unexpected error occurred while creating a procedure using the initializeProc() function.

User response: Check the QMFwin API logs to find the root error.

API1016  A new query could not be created. There may be too many documents already open, or there may be insufficient memory.
Explanation: An unexpected error occurred while creating a query using the initializeQuery() function.

User response: Check the QMFwin API logs to find the root error.

API1024  The specified column delimiter (0) is not valid.
Explanation: An invalid value was specified for the ColumnDelimiter parameter of an export() function.

User response: Specify a valid value for the ColumnDelimiter parameter of any export() function.

API1026  The specified file name is not valid.
Explanation: An invalid or empty value was specified for the FileName parameter of an export() function.

User response: Specify a valid value for the FileName parameter of any export() function.

API1027  The specified format (0) is not valid.
Explanation: An invalid value was specified for the Format parameter in an export() function.

User response: Specify a valid value for the Format parameter of any export() function.

API1028  The specified form name is not valid.
Explanation: An invalid or empty value was specified for the Form parameter of a report related function.

User response: Specify a valid value for the Form parameter of any report related functions.

API1029  The specified form source (0) is not valid.
Explanation: An invalid value was specified for the SourceType parameter of a report related function.

User response: Specify a valid value for the SourceType parameter of any report related function.

API1030  The global variable name is invalid.
Explanation: An invalid value was specified for the GlobalVariableName parameter of a global variables related function.

User response: Specify a valid value for the
GlobalVariableName parameter of any global variables related function.

API1031 The specified host variable is not valid.
Explanation: An invalid value was specified for the Value parameter of a setHostVariable() function.
User response: Specify a valid value for the Value parameter of any setHostVariable() function.

API1033 The specified owner and name ({{0}}) are not valid.
Explanation: An invalid value was specified for the OwnerAndName parameter of an API function.
User response: Specify a valid value for the OwnerAndName parameter of any relevant API function.

API1034 The specified page length ({{0}}) is not valid.
Explanation: An invalid value was specified for the PageLength parameter of an exportReport() or exportVisualReport() function.

API1035 The specified page width ({{0}}) is not valid.
Explanation: An invalid value was specified for the PageWidth parameter of an exportReport() or exportVisualReport() function.

API1037 The specified procedure ID ({{0}}) is not valid.
Explanation: An invalid value was specified for the ProcID parameter or a procedure related function.
User response: Specify a valid value for the ProcID parameter of any procedure related functions.

API1038 The specified resource ({{0}}) is not valid.
Explanation: An invalid value was specified for the Resource parameter of a getResourceLimit() function.

API1039 The specified range of rows and columns is not valid.
Explanation: Invalid values were specified for the FirstRow, FirstCol, LastRow, and LastCol parameters of an exporting result set function.
User response: Specify valid values for the FirstRow, FirstCol, LastRow, and LastCol parameters of any exporting result set function.

API1040 The specified string delimiter ({{0}}) is not valid.
Explanation: An invalid value was specified for the StringDelimiter parameter of an export() function.
User response: Specify a valid value for the StringDelimiter parameter of any export() function.

API1041 The specified table name is not valid.
Explanation: An invalid value was specified for the TableName parameter of an export() function.
User response: Specify a valid value for the TableName parameter of any export() function.

API1042 The specified target server is invalid.
Explanation: An invalid value was specified for the Server parameter of a saveData() or copyQMFObject() function.
User response: Specify a valid value for the Server parameter of any saveData() or copyQMFObject() function.

API1044 The specified time ({{0}}) is not valid.
Explanation: An invalid value was specified for the Time parameter of a getQMFObjectInfo() function.
User response: Specify a valid value for the Time parameter of any getQMFObjectInfo() function.

API1045 The specified type ({{0}}) is not valid.
Explanation: An invalid value was specified for the Type parameter of a getQMFObjectInfo() function.
User response: Specify a valid value for the Type parameter of any getQMFObjectInfo() function.

API1046 The specified variable ({{0}}) is not valid.
Explanation: An invalid value was specified for the VariableName parameter of a setVariable() function.
User response: Specify a valid value for the VariableName parameter of any setVariable() function.
### API1053: SQL Verb Cannot Be Used with ExecuteStoredProcedure()

**Explanation:** A query that does not contain the CALL SQL verb was specified for an executeStoredProcedureEx() function.

**User response:** Specify a CALL query for any executeStoredProcedureEx() function.

### API1054: Can't Prepare a Static Section.

**Explanation:** A static query was specified for a prepare() function; only dynamic queries can be specified.

**User response:** Specify a dynamic query for any prepare() function.

### API10004: The Values Specified for ForceDialog and SuppressDialog Are Mutually Exclusive.

**Explanation:** The initializeServer() function was called and the values that were specified for ForceDialog and SuppressDialog parameters were mutually exclusive.

**User response:** Check the code that invokes the initializeServer() function.

### API10005: The Specified Value Is Not Valid.

**Explanation:** The value that was specified for the parameter of a function is incorrect.

**User response:** Check to ensure that all of the specified parameter values for all of the functions that were called in the API instance are valid.

### BIC00000: Internal Error

**Explanation:** There was an internal error.

**User response:** Collect support data by selecting Help > Collect Support Data and forward it to customer support.

### BIC00007: Object Is Not Supported by DataQuant

**Explanation:** The specified object is not supported by DataQuant.

**User response:** No action is required.

### BIC00003: Fetching of Children Is Cancelled by User.

**Explanation:** The operation was cancelled by the user.

**User response:** Retry to fetch the specified objects.

### CLC00002: Unable to Evaluate LinReg: Data Series Have Different Numbers of Elements.

**Explanation:** The data series specified for the x and y variables contain different numbers of data points.

**User response:** Specify data series that contain the same number of data points.

### CLC00003: Unable to Evaluate LinReg: Not Enough Data.

**Explanation:** The data series specified do not contain enough rows to calculate the linear regression.

**User response:** Specify a query result set that contains more than one row of data points.
CLC00004 Unable to evaluate linReg: coefficient is out of range.
Explanation: The value that you specified for the nParam parameter of the linreg() function is invalid.
User response: Specify either a 0 or a 1 for the nParam parameter. A 0 returns the slope of the linear regression. A 1 returns the intercept of the linear regression.

CLC00005 Illegal data row range specified: from {0} to {1}
Explanation: The range of data rows specified for the nFirstRow and nLastRow parameters is invalid.
User response: Specify a valid range of data rows. The nFirstRow value must specify a row before the nLastRow value in the query result set.

CLC00006 Unable to calculate gammaIP: domain error.
Explanation: Invalid arguments were specified in the gammaIP function.
User response: Specify a valid argument in the gammaIP function.

CLC00009 "{0}" function is not found in Lib class, but supported in expressions.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

CLC00010 "{0}" function has wrong arguments description.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

CLC00011 Type conversion error. Object of {0} type cannot be converted to {1} type.
Explanation: The indicated object can not be converted to the indicated type.
User response: Specify a valid object to be converted to the indicated type.

CLC00012 Evaluation of expression failed: {0}
Explanation: The evaluation of the indicated expression failed.
User response: Specify a valid expression for evaluation.

CLC00014 Evaluation of expression has failed.
Explanation: The evaluation of the specified expression has failed.
User response: Check the syntax of the expression for errors and try again.

CLC00016 Cannot parse an expression: {0}
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

CLC00018 Unknown function name: {0}.
Explanation: The indicated function name is not valid.
User response: Specify a valid function name.

CLC00030 Unable to process URL: "{0}".
Explanation: The indicated URL is invalid.
User response: Specify a valid URL.

CLC00034 "{0}" function description is missing.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

CLC00038 Multiple declarations of "{0}" library in LibraryFactory class.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

CLC00039 Multiple declarations of "{0}" library in LibraryFactory class.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

CLC00040 Cannot load "{0}" library in LibraryFactory class.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.
Types "[0]" and "[1]" are not acceptable for "[2]" function.

Explanation: The arguments that were passed to the function are of invalid types.

User response: Check the syntax of the expression for errors and try again.

Type "[0]" is not acceptable for "[1]" function.

Explanation: The specified function does not accept parameters of the given type.

User response: Check the syntax of the expression for errors and try again.

Information service is undefined.

Explanation: An internal error occurred.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

Aggregation service is undefined.

Explanation: An internal error occurred.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

Hierarchy service is undefined.

Explanation: An internal error occurred.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

Color service is undefined.

Explanation: An internal error occurred.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

PPI service is undefined.

Explanation: An internal error occurred.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

Value provider is undefined.

Explanation: An internal error occurred.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

> Collect Support Data and forward it to customer support.

Function "[0]" is defined for [1] argument(s).

Explanation: An invalid number of parameters were specified for the indicated function.

User response: Specify a correct number of parameters for the indicated function.

Function "[0]" is undefined for argument [1].

Explanation: An invalid number of parameters were specified for the indicated function.

User response: Specify a valid number of parameters for the indicated function.

One of the operands does not have measurement units.

Explanation: One of the operands of the function does not have measurement units.

User response: Make sure that all of the operands in the function have measurement units.

Cannot multiply two measured values.

Explanation: Measured values cannot be multiplied.

User response: Specify values that are not measured to be multiplied.

Cannot create list from elements of "[0]" type.

Explanation: A list cannot be created using elements of the indicated type.

User response: Specify elements of a different type to be listed.

List cannot contain elements of different types.

Explanation: The specified list cannot contain elements of different types.

User response: Specify elements of the same type to be listed.

Types "[0]", "[1]" and "[2]" are not acceptable for "[3]" function.

Explanation: The indicated function is not compatible with the given types.

User response: Check the syntax of the expression for errors and try again.
Argument(s) with measurement units are not acceptable for "(0)" function.

Explanation: The indicated function can not contain arguments with measurement units.

User response: Remove all measurement units from the arguments of the function.

Incorrect index given.

Explanation: An error occurred in the LineText function.

User response: Check the arguments that are specified for the LineText function for errors.

Types "(0)", "(1)", "(2)" and "(3)" are not acceptable for "(4)" function.

Explanation: The indicated function is not compatible with the given types.

User response: Check the expression syntax for errors and try again.

Value for variable "(0)" is not defined.

Explanation: The value for the indicated variable is not defined.

User response: Define the value for the indicated variable.

Function "(0)" caused error: [1]

Explanation: The indicated function caused the indicated error.

User response: Correct the indicated error and rerun the function.

Property "(0)" contains error: [1]

Explanation: The indicated property caused the indicated error.

User response: Correct the indicated error and rerun the function.

List sizes are not comparable.

Explanation: Invalid parameters with type List were specified for add, multiply, divide, or subtract functions.

User response: Specify a valid value for the List parameter of any add, multiply, divide, or subtract functions.

Cannot be evaluated for empty list.

Explanation: No value was specified for the List parameter of a function.

User response: Specify a valid value for the List parameter of any relevant function.

Function @if has odd number of parameters: @if(cond1, exp1, cond2, exp2, ..., expn). expn is used when there aren't true conditions

Explanation: The syntax of an @if function is invalid.

User response: Check the syntax of any @if functions for errors and try again.

The feature is not supported.

Explanation: The specified feature is not supported by the database.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

No column named "(0)" was found.

Explanation: The indicated column was not found.

User response: Check the column name for errors or specify a different column name.

The current row is unsuitable for get/set operations.

Explanation: The specified row can not be retrieved or set.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

The database cursor has been closed.

Explanation: The requested operation could not be completed because the cursor is closed.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

An error occurred while accessing the file. [0]

Explanation: An input/output error occurred while accessing the file.

User response: Collect support data by selecting Help
> Collect Support Data and forward it to customer support.

DAL01007  Character encoding [0] is not supported.
Explanation:  The indicated character encoding is not supported.
User response:  Use a character set that is supported by your current JVM.

DAL01008  An error occurred while accessing the database.
Explanation:  An error occurred while accessing the indicated database.
User response:  Collect support data by selecting Help
> Collect Support Data and forward it to customer support.

DAL01009  An internal error occurred. [0]
Explanation:  The indicated internal error occurred.
User response:  Collect support data by selecting Help
> Collect Support Data and forward it to customer support.

DAL01011  Class [0] is not suitable for server description
Explanation:  An internal error occurred.
User response:  Collect support data by selecting Help
> Collect Support Data and forward it to customer support.

DAL01012  Requested operation is not available in offline mode
Explanation:  The specified operation is not available in offline mode.
User response:  Specify a different operation to apply to the active object, or switch to online mode and try again.

DAL01013  Cannot create connection to server "[0]" due to the following: [1]
Explanation:  A connection to the indicated server could not be established due to the indicated error.
User response:  Resolve the indicated error and retry to connect to the server.

DAL01014  Invalid set of connection parameters used,
Explanation:  An internal error occurred.
User response:  Collect support data by selecting Help
> Collect Support Data and forward it to customer support.

DAL01016  Cannot parse data (base64 encoding rules violation)
Explanation:  The XML file is corrupted.
User response:  Fix any errors in the XML file and try again.

DAL01017  Duplicate query with name "[0]" is detected in package "[1]".
Explanation:  An internal error occurred; the indicated package contains several queries with the same specified name.
User response:  Collect support data by selecting Help
> Collect Support Data and forward it to customer support.

DAL01018  The source data is not compatible with the target table.
Explanation:  The attempt to save data in the database table failed because of the following:
- Number of columns does not match.
- Columns have incompatible types.
User response:  Select compatible target table and try again.

DAL01019  Operation is not supported.
Explanation:  An internal error occurred; the requested operation can not be performed due to internal limitations.
User response:  Collect support data by selecting Help
> Collect Support Data and forward it to customer support.

DAL01020  Invalid SQL type definition: "[0]"
Explanation:  An internal error occurred; the indicated SQL type definition is invalid (it can be mapped to more than one known data types).
User response:  Collect support data by selecting Help
> Collect Support Data and forward it to customer support.

DAL01021  This product does not contain SQLj support module
Explanation:  DataQuant does not support or can not initialize the SQLj support module.
User response:  Collect support data by selecting Help
> Collect Support Data and forward it to customer support.
DAL01022 Cannot have two different driver libraries with the same driver name.
Explanation: The driver with the specified driver name is already defined in another library definition.
User response: Use (edit) the already defined driver library instead of creating a new one.

DAL01023 An error has occurred while saving driver libraries list: {0}
Explanation: The indicated error occurred while the driver libraries list was being saved.
User response: Resolve the indicated error and try again.

DAL01025 An error has occurred while loading the driver libraries list: {0}
Explanation: The indicated error occurred while trying to load the driver libraries list.
User response: Resolve the indicated error and try again.

DAL01026 Operation cannot be performed due to resource limits restriction: {0}
Explanation: The operation could not be performed due to the indicated resource limits restriction.
User response: Check your current resource limits by selecting Resource Limits from the View menu. Specify an operation within your resource limits or contact your administrator for specific permissions.

DAL01027 User name is missing.
Explanation: The user name is missing from the User Information dialog name field.
User response: Specify a user name.

DAL01028 Password is missing.
Explanation: The password is missing from the User Information dialog.
User response: Specify a password.

DAL01029 Wrong JDBC URL: {0}.
Explanation: The indicated JDBC URL cannot be processed by DataQuant. The URL may be incorrect.
User response: Check the JDBC URL for errors or specify a different URL.

DAL01030 Query parameter {0} is of wrong Java type. Data type "{1}" is required.
Explanation: An internal error occurred while running the query. The query parameters have invalid types.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01031 Cached server descriptor cannot be restored due to the following: {0}
Explanation: The cached server descriptor cannot be restored due to the following error.
User response: Resolve the indicated error and try again.

DAL01032 An error occurred while making remote method call. {0}
Explanation: The indicated error occurred while the operation was running on the server side.
User response: Refer to the indicated error description for possible response(s).

DAL01033 Client request cannot be processed by remote server.
Explanation: An internal error occurred; the client request cannot be processed by a remote server.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01034 Attempt to use incomplete metadata descriptor is detected.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01035 Editing is not supported for this data set.
Explanation: An internal error occurred; this data set cannot be edited.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01036 The current row is unsuitable for editing.
Explanation: An internal error occurred; the current row can not be edited.
User response: Collect support data by selecting Help
> Collect Support Data and forward it to customer support.

DAL01037  The database server does not support table renaming
Explanation: Tables in this database server can not be renamed by DataQuant.
User response: No action is required.

DAL01038  The edit operation was cancelled.
Explanation: The table editing operation was cancelled by the user.
User response: No action is required.

DAL01039  The record could not be updated (possibly concurrent edit/delete operations).
Explanation: The record could not be updated. This is most likely because it is locked due to a concurrent edit/delete operation.
User response: Refresh the result set and try again.

DAL01040  The record could not be deleted (possibly concurrent edit/delete operations).
Explanation: The record could not be deleted. This is most likely because it is locked due to a concurrent edit/delete operation.
User response: Refresh the result set and try again.

DAL01041  The value of a calculated column is longer than specified in the metadata
Explanation: The data could not be exported because one or more of the calculated columns in the result set contains data that is longer than what is allowed in the column's definition.
User response: Increase the maximum column length or modify the data in the calculated column.

DAL01042  Invalid type conversion detected.
Explanation: An internal error occurred; a type conversion in the specified operation is invalid.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01043  An attempt was made to modify a deleted record.
Explanation: An internal error occurred; records that have already been deleted can not be modified or restored.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01044  Connection is blocked due to the following: {0}
Explanation: The attempted connection was blocked due to the indicated error or restriction.
User response: Depending on the indicated problem, either resolve the specified error or contact your administrator to remove the restriction.

DAL01045  This product does not contain static mode support module for selected database.
Explanation: This product does not contain static mode support module for selected database.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01046  Driver class name "{0}" is not loaded. This may be caused by an incorrect driver class name or missing JDBC driver libraries.
Explanation: The indicated driver class name can not be loaded. This may be caused by an incorrect driver class name or missing JDBC driver libraries.
User response: Check that there is a proper definition for the JDBC driver library that refers to the indicated driver class.

DAL01049  Selected protection mode is not supported by database.
Explanation: The selected protection mode is not supported by the specified database.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01050  Driver specific functionality is not available due to: {0}
Explanation: Driver specific functionality is not available due to the indicated error.
User response: Resolve the indicated error and retry. Verify that the JDBC driver library for the driver is
properly configured and that it contains all of the necessary JAR files. If the problem persists, collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01200  Generic data access error has occurred.  

Explanation: An error with no textual description occurred. Additional information may follow the message. This error code may refer to different error conditions depending on product versions.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01201  Generic data access error has occurred.  

Explanation: An error with no textual description occurred. Additional information may follow the message. This error code may refer to different error conditions depending on product versions.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01202  Generic data access error has occurred.  

Explanation: An error with no textual description occurred. Additional information may follow the message. This error code may refer to different error conditions depending on product versions.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01203  Generic data access error has occurred.  

Explanation: An error with no textual description occurred. Additional information may follow the message. This error code may refer to different error conditions depending on product versions.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01204  Generic data access error has occurred.  

Explanation: An error with no textual description occurred. Additional information may follow the message. This error code may refer to different error conditions depending on product versions.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01205  Generic data access error has occurred.  

Explanation: An error with no textual description occurred. Additional information may follow the message. This error code may refer to different error conditions depending on product versions.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01206  Generic data access error has occurred.  

Explanation: An error with no textual description occurred. Additional information may follow the message. This error code may refer to different error conditions depending on product versions.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01207  Generic data access error has occurred.  

Explanation: An error with no textual description occurred. Additional information may follow the message. This error code may refer to different error conditions depending on product versions.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DAL01208  Generic data access error has occurred.  

Explanation: An error with no textual description occurred. Additional information may follow the message. This error code may refer to different error conditions depending on product versions.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

DDM0001  Unexpected error occurred.  

Explanation: An internal error in the driver code was detected.

User response: Send an error report to the developers.
<table>
<thead>
<tr>
<th>Code</th>
<th>Message</th>
<th>Explanation</th>
<th>User response</th>
</tr>
</thead>
<tbody>
<tr>
<td>DDM0002</td>
<td>Object {0} have no fields with codepoint {1}</td>
<td>An internal error in the driver code was detected.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0003</td>
<td>Socket open exception. {0}</td>
<td>Unable to establish TCP/IP connection.</td>
<td>Check the host and port details in the JDBC URL; ping the server to check that the network connection works.</td>
</tr>
<tr>
<td>DDM0004</td>
<td>Socket write exception. {0}</td>
<td>There was a network connection failure or an internal error in the driver code was detected.</td>
<td>Check your network connection. If the connection works fine, send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0005</td>
<td>Data format exception detected.</td>
<td>An internal error in the driver code was detected.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0006</td>
<td>Invalid JDBC URL {0}</td>
<td>An internal error in the driver code was detected.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0007</td>
<td>DSS header format error detected.</td>
<td>An internal error in the driver code was detected.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0008</td>
<td>Unsupported CCSID {0}</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0009</td>
<td>Cipher error {0}</td>
<td>An internal error in the driver code was detected.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0010</td>
<td>Object {0} does not contain required field {1}</td>
<td>An internal error in the driver code was detected.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0011</td>
<td>Object {0} does not support value {1}</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0012</td>
<td>Syntax error detected: {0}</td>
<td>An internal error in the driver code was detected.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0013</td>
<td>Syntax error detected: {0}</td>
<td>An internal error in the driver code was detected.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0014</td>
<td>Object {0} can hold no more repeated fields {1}</td>
<td>An internal error in the driver code was detected.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0015</td>
<td>Illegal port number value: {0}</td>
<td>The JDBC URL contains an invalid value in the Port field.</td>
<td>Specify a correct port number in the JDBC URL.</td>
</tr>
<tr>
<td>DDM0016</td>
<td>{0} is invalid length for the field {1} of object {2}</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0017</td>
<td>Unknown FD:OCA descriptor type id {0}</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>Code</td>
<td>Message</td>
<td>Explanation</td>
<td>User response</td>
</tr>
<tr>
<td>----------</td>
<td>--------------------------------------------------------------------------</td>
<td>------------------------------------------------------------------------------</td>
<td>---------------</td>
</tr>
<tr>
<td>DDM0018</td>
<td>Null value for non nullable data detected.</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0019</td>
<td>Whole number of FD:OCA [0] triplet repeating groups cannot be placed in [1] bytes</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0020</td>
<td>Illegal CPT triplet detected.</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0021</td>
<td>Illegal duplicated field [1] in object [0]</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0022</td>
<td>Referenced FD:OCA triplet [0] not found</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0023</td>
<td>FD:OCA [0] triplets cannot be referenced from RLO triplet</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0024</td>
<td>Replication factor [0] is detected in RLO triplet while only 1 and 0 is allowed by DRDA</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0025</td>
<td>FD:OCA [0] triplets cannot be referenced from GDA triplet</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0026</td>
<td>Unknown DRDA type [0]</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0027</td>
<td>Infinite element count is not allowed for RLO handler of [0]</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0028</td>
<td>An error occurred while parsing triplet handler for DRDA type [0]</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0029</td>
<td>Unspecified value for JDBC statement variable [0]</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0030</td>
<td>Illegal type conversion.</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0031</td>
<td>Illegal date or time format detected: [0]</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0032</td>
<td>Prepared statement parameter type [0] (from java.sql.Type set) is not supported.</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>DDM0033</td>
<td>Unknown DB2 type [0]</td>
<td>An internal error was detected in the driver code.</td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>
### DDM0035 Statement is closed.
**Explanation:** An attempt to work with a closed Statement object was detected.
**User response:** Fix your program code.

### DDM0036 ResultSet is closed.
**Explanation:** An attempt to work with a closed ResultSet object was detected.
**User response:** Fix your program code.

### DDM0037 Object is closed.
**Explanation:** An attempt to work with a closed object was detected.
**User response:** Fix your program code.

### DDM0038 Operation cannot be performed in current ResultSet state.
**Explanation:** An attempt to work with a ResultSet column of a result set that is not positioned on any valid record was detected.
**User response:** Fix your program code.

### DDM0039 Error reply is received.
**Explanation:** An internal error in the driver code was detected.
**User response:** Send an error report to the developers.

### DDM0040 Error reply is received for [0] command.
**Explanation:** An internal error in the driver code was detected.
**User response:** Send an error report to the developers.

### DDM0041 Error detected while processing [0] object.
**Explanation:** An internal error in the driver code was detected.
**User response:** Send an error report to the developers.

### DRL00001 The maximum number of connections to the server has been reached.
**Explanation:** The maximum number of connections to the server has been reached.
**User response:** Increase the maximum number of connections parameter in the Repository Storage dialog or set the value to zero to remove this restriction.

### DSCERRCD0001 FD:OCA triplet is not used in DRDA descriptors, or the type code is invalid.
**Explanation:** An internal error in the driver code was detected.
**User response:** Send an error report to the developers.

### DSCERRCD0002 FD:OCA triplet sequence error.
**Explanation:** An internal error in the driver code was detected.
**User response:** Send an error report to the developers.

### DSCERRCD0003 An array description is required, and this is not one (too many or too few Row Lay Out (RLO) triplets).
**Explanation:** An internal error in the driver code was detected.
**User response:** Send an error report to the developers.

### DSCERRCD0004 A row description is required, and this is not one (too many or too few RLO triplets).
**Explanation:** An internal error in the driver code was detected.
**User response:** Send an error report to the developers.

### DSCERRCD0005 Late environmental descriptor just received, not supported.
**Explanation:** An internal error in the driver code was detected.
**User response:** Send an error report to the developers.

### DSCERRCD0006 Malformed triplet; required parameter is missing.
**Explanation:** An internal error in the driver code was detected.
**User response:** Send an error report to the developers.

### DSCERRCD0007 Parameter value is not acceptable.
**Explanation:** An internal error in the driver code was detected.
**User response:** Send an error report to the developers.

### DSCERRCD0017 Meta-Data Descriptor (MDD) present is not recognized as a Structured Query Language (SQL) descriptor.
**Explanation:** An internal error in the driver code was detected.
<table>
<thead>
<tr>
<th>Error Code</th>
<th>Message Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DSCERRCD0018</td>
<td>MDD class is not recognized as a valid SQL class.</td>
</tr>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

| DSCERRCD0019 | MDD type is not recognized as a valid SQL type. |
| **Explanation:** | An internal error in the driver code was detected. |
| **User response:** | Send an error report to the developers. |

| DSCERRCD0033 | Representation is incompatible with SQL type (in prior MDD). |
| **Explanation:** | An internal error in the driver code was detected. |
| **User response:** | Send an error report to the developers. |

| DSCERRCD0034 | CCSID is not supported. |
| **Explanation:** | An internal error in the driver code was detected. |
| **User response:** | Send an error report to the developers. |

| DSCERRCD0050 | Group Data Array (GDA) references a local identifier (LID) which is not a Scalar Data Array (SDA) or GDA. |
| **Explanation:** | An internal error in the driver code was detected. |
| **User response:** | Send an error report to the developers. |

| DSCERRCD0051 | GDA length override exceeds limits. |
| **Explanation:** | An internal error in the driver code was detected. |
| **User response:** | Send an error report to the developers. |

| DSCERRCD0052 | GDA precision exceeds limits. |
| **Explanation:** | An internal error in the driver code was detected. |
| **User response:** | Send an error report to the developers. |

<p>| DSCERRCD0053 | GDA scale greater than precision or scale negative. |
| <strong>Explanation:</strong> | An internal error in the driver code was detected. |
| <strong>User response:</strong> | Send an error report to the developers. |</p>
<table>
<thead>
<tr>
<th>ERM0006</th>
<th>Manager-level Conflict.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0007</th>
<th>Conversational Protocol Error.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0008</th>
<th>Parameter Not Supported.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0009</th>
<th>Resource Limits Reached.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0010</th>
<th>Data Stream Syntax Error.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0011</th>
<th>Target Not Supported.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0012</th>
<th>Parameter Value Not Supported.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0013</th>
<th>Conversational Protocol Error.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0014</th>
<th>RDB Currently Accessed.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0015</th>
<th>RDB Access Failed Reply Message.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0016</th>
<th>Not Authorized To RDB.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0017</th>
<th>RDB Not Found.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0018</th>
<th>Resource Limits Reached.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0019</th>
<th>Data Stream Syntax Error.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0020</th>
<th>Invalid Request.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0021</th>
<th>Abnormal End Unit of Work Condition.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0022</th>
<th>RDB Package Binding Process Active.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>ERM0023</th>
<th>Query Not Open.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error in the driver code was detected.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Send an error report to the developers.</td>
</tr>
</tbody>
</table>
ERM0024 • RDB Not Accessed.
Explanation: An internal error in the driver code was detected.
User response: Send an error report to the developers.

ERM0025 • Data Descriptor Mismatch.
Explanation: An internal error in the driver code was detected.
User response: Send an error report to the developers.

ERM0026 • Object Not Supported.
Explanation: An internal error in the driver code was detected.
User response: Send an error report to the developers.

ERM0027 • RDB Package Binding Not Active.
Explanation: An internal error in the driver code was detected.
User response: Send an error report to the developers.

ERM0028 • RDB Update Reply Message.
Explanation: An internal error in the driver code was detected.
User response: Send an error report to the developers.

ERM0029 • SQL Error Condition.
Explanation: An internal error in the driver code was detected.
User response: Send an error report to the developers.

ERM0030 • End of Data.
Explanation: An internal error in the driver code was detected.
User response: Send an error report to the developers.

ERM0031 • End of Query.
Explanation: An internal error in the driver code was detected.
User response: Send an error report to the developers.

ERM0032 • Commitment Request.
Explanation: An internal error in the driver code was detected.
User response: Send an error report to the developers.

ERM0033 • End Unit of Work Condition.
Explanation: An internal error in the driver code was detected.
User response: Send an error report to the developers.

ERM0034 • Invalid Descriptor.
Explanation: An internal error in the driver code was detected.
User response: Send an error report to the developers.

ERM0035 • Open Query Complete.
Explanation: An internal error in the driver code was detected.
User response: Send an error report to the developers.

ERM0036 • Open Query Failure.
Explanation: An internal error in the driver code was detected.
User response: Send an error report to the developers.

ERM0037 • Query Previously Opened.
Explanation: An internal error in the driver code was detected.
User response: Send an error report to the developers.

ERM0038 • Command Violation.
Explanation: An internal error in the driver code was detected.
User response: Send an error report to the developers.

ERM1200 • Generic DRDA error has occurred. 0
Explanation: The indicated generic DRDA error has occurred.
User response: Resolve the indicated generic DRDA error and try again.

ERM1201 • Generic DRDA error has occurred. 0, 1
Explanation: The two indicated generic DRDA error have occurred.
User response: Resolve the two indicated errors and try again.
ERM1202  Generic DRDA error has occurred. [0], [1], [2]
Explanation: The three indicated generic DRDA errors have occurred.
User response: Resolve the three indicated errors and try again.

ERM1203  Generic DRDA error has occurred. [0], [1], [2], [3]
Explanation: The four indicated generic DRDA errors have occurred.
User response: Resolve the four indicated errors and try again.

ERM1204  Generic DRDA error has occurred. [0], [1], [2], [3], [4]
Explanation: The five indicated generic DRDA errors have occurred.
User response: Resolve the five indicated errors and try again.

ERM1205  Generic DRDA error has occurred. [0], [1], [2], [3], [4], [5]
Explanation: The six indicated generic DRDA errors have occurred.
User response: Resolve the six indicated errors and try again.

ERM1206  Generic DRDA error has occurred. [0], [1], [2], [3], [4], [5], [6]
Explanation: The seven indicated generic DRDA errors have occurred.
User response: Resolve the seven indicated errors and try again.

ERM1207  Generic DRDA error has occurred. [0], [1], [2], [3], [4], [5], [6], [7]
Explanation: The eight indicated generic DRDA errors have occurred.
User response: Resolve the eight indicated errors and try again.

ERM1208  Generic DRDA error has occurred. [0], [1], [2], [3], [4], [5], [6], [7], [8]
Explanation: The nine indicated generic DRDA errors have occurred.
User response: Resolve the nine indicated errors and try again.

EXPT1000  Cannot perform export command: [0]
Explanation: The indicated export command can not be performed.
User response: Resolve the first indicated error and try again.

EXPT1001  An unsupported data type [0] was encountered.
Explanation: The result set contains columns with the types that are not supported by the IXF370 format.
User response: Make sure that the result set does not contain columns with any of the following types: BINARY, VARBINARY, ROWID, LONGVARBINARY, CLOB, DBCLOB, BLOB, GRAPHIC, VARGRAPHIC, or LONGVARGRAPHIC.

EXPT1002  The length of column [0] must be less than [1].
Explanation: The length of the indicated column must be less than the indicated length due to restrictions of the IXF format.
User response: Make sure that the indicated data column is less than the indicated length. Try to export using another format (for example, XML).

EXPT1003  The precision of column [0] must be an odd number between [1] and [2], inclusive.
Explanation: The precision of the indicated column must be an odd number between the two indicated values due to restrictions of the IXF format.
User response: Make sure the precision of the indicated data column is an odd number between the two indicated values. Try to export using another format (for example, XML).

EXPT1004  The data could not be exported due to an unsupported CCSID [0].
Explanation: The indicated CCSID is not supported for export.
User response: Specify a valid CCSID for export. A list of supported CCSIDs can be obtained from the CCSID drop down menu in the Export dialog.

EXPT1005  The data could not be exported because the single-byte code page is not specified.
Explanation: A result set containing CLOB, CHAR, VARCHAR, or LONGVARCHAR columns was exported in IXF format but a CCSID that contains a single-byte code page was not specified.
User response: Select a CCSID that contains a single-byte code page.

EXPT1006 The data could not be exported because the double-byte code page is not specified.

Explanation: A result set containing DBCLOB, GRAPHIC, VARGRAPHIC, or LONGVARGRAPHIC columns was exported in IXF format but a CCSID that contains a double-byte code page was not specified.

User response: Select a CCSID that contains a double-byte code page.

EXPT1007 The value in column [0] exceeds the maximum length supported by the dBASE file format.

Explanation: The value in the indicated column exceeds the character limit that is supported by the dBASE file format.

User response: Shorten the value in the indicated column or specify a different file format for export.

EXPT1008 The data type of column [0] is not supported in dBASE files.

Explanation: The data type of the indicated column is not supported by the dBASE file format.

User response: Specify a different data type for the indicated column or select a different file format for export.

EXPT1009 The row length in a dBASE file cannot exceed 4000 characters.

Explanation: The summary length of all of the result set’s columns exceeds 4,000 characters, which is not supported by the dBASE file format.

User response: Make sure that the length is shorter than 4,000 characters, or select a different file format for export.

EXPT1010 The specified data source does not exist.

Explanation: There was an attempt to export a result set to a data source that does not exist.

User response: Specify a valid data source.

EXPT1011 The operation cancelled.

Explanation: The user cancelled the export command when prompted for a user name and password.

User response: Specify a valid user name and password for the export command.

EXPT1012 Name must be specified.

Explanation: There was an attempt to export a result set to a table that does not exist.

User response: Specify a valid export table.

EXPT1013 Invalid storage entry.

Explanation: The storage file contains corrupted data.

User response: Setup fonts and font mapping for PDF export.

EXPT1014 Cannot create font {0} for PDF

Explanation: The indicated font can not be created for the PDF file format.

User response: Specify a different font for conversion, specify a different file format for export, or set up fonts and font mapping for PDF file export.

EXPT1015 {0} font is not mapped

Explanation: The indicated font name is not mapped to the physical font file.

User response: Setup fonts and font mapping for PDF file export.

FRM1001 The input line type is invalid.

Explanation: Corrupted form. The form loader is hanging on a line with a certain first character, because every line in a form file, according to the form file format, starts with certain character (T, V, R, H, E, *) that define the line type. The current line’s first character has an unknown type.

User response: Try to open and save again with exact same version of DataQuant that was used to create the form.

FRM1002 The input line is too short.

Explanation: The form loader is hanging on a line with a certain character count, because each line type must have a character count greater than a certain number (T > 15, V > 11, R > 3, or H > 44). The current line is too short.

User response: Try to open and save again with exact same version of DataQuant that was used to create the form.

FRM1003 The boolean value is invalid. Valid values are YES or NO.

Explanation: Corrupted form: A boolean value in the form file has a string other than YES or NO.

User response: Try to open and save again with exact
same version of DataQuant that was used to create the form.

FRM1004  The numeric value is invalid. Valid values are NONE, COLUMNS, DEFAULT, or an integer value.

Explanation: Corrupted form: The form loader waits for a valid Integer value or predefined word (NONE, COLUMNS, or DEFAULT).

User response: Try to open and save again with exact same version of DataQuant that was used to create the form.

FRM1005  An invalid code was encountered.

Explanation: Corrupted form: Every table structure in a form file has its own column types. The form loader waits for certain column types depending on the table type.

User response: Try to open and save again with exact same version of DataQuant that was used to create the form.

FRM1006  The object header is too short.

Explanation: The form loader is hanging on a line with a certain character count, because each line type must have a character count greater than a certain number (T > 15, V > 11, R > 3, or H > 44). The current line is too short.

User response: Try to open and save again with exact same version of DataQuant that was used to create the form.

FRM1007  The object header is invalid.

Explanation: Corrupted form: The object header contains unexpected fields or data.

User response: Try to open and save again with exact same version of DataQuant that was used to create the form.

FRM1008  Invalid usage code.

Explanation: The “new column” of the form dialog contains an invalid usage code.

User response: Leave the usage code field empty or select one from the combo box.

FRM1009  The value specified for {0} is invalid. Valid values are {1}.

Explanation: In the form dialog page, there are several places where you can type or choose a listed value. An invalid listed value was entered.

User response: Enter a valid value from the list.

FRM1010  The column heading "[0]" is too long.

Explanation: The length of the column heading must be less than 40 characters.

User response: Ensure that the column heading is less than 40 characters.

FRM1011  There is no condition expression with ID {0}.

Explanation: A form has a few detail blocks in the “detail” branch (by default – one) and each detail can use a condition that defines the detail as enabled or disabled. The condition typed in the C1, C2, .. C999 format and the condition with the according index should exist in the “conditions” branch.

User response: Specify YES, NO, or Ci where Ci is the existing condition.

FRM1012  Line Wrapping Width must be NONE when Number of Fixed Columns is not NONE.

Explanation: In the Options page of the form designer, when any number is defined in the Number of Fixed Columns field, the Line Wrapping Width field must be set to NONE.

User response: Specify NONE in either the Line Wrapping Width field or the Number of Fixed Columns field.

FRM1013  The ID for calculation {0} is already being used by a different calculation.

Explanation: In the Calculations page of the form designer, two or more calculations with the same ID were specified.

User response: Ensure that each calculation ID is unique.

FRM1014  The ID for condition {0} is already being used by a different condition.

Explanation: In the Conditions page of the form designer, two or more conditions with the same ID were specified.

User response: Ensure that each of the conditions are unique.

FRM1015  Line Wrapping Width must be 0 if any column uses a column wrapping edit code.

Explanation: If a column uses the wrapping edit code, then in the Options page of the form designer, the Line Wrapping Width field must be set to NONE.

User response: Either set the Line Wrapping Width
field to NONE, or do not use the line wrapping edit code.

FRM1016 Automatically Reorder Report Columns must be TRUE when using the ACROSS usage code.

Explanation: If any columns have the ACROSS usage code, then in the Options page of the form designer, the Automatically Reorder Report Columns field must be set to TRUE.

User response: Either set the Automatically Reorder Report Columns field to TRUE or do not use the ACROSS usage code.

FRM1017 The ACROSS usage code requires other columns to use the GROUP and aggregate usage codes.

Explanation: If any columns use the ACROSS usage code, then all other columns must use either the OMIT, GROUP, or AGGREGATION usage code.

User response: Set the OMIT, GROUP, or AGGREGATION usage codes for all other columns.

FRM1018 Only one column can use the ACROSS usage code.

Explanation: Only one column can use the ACROSS usage code.

User response: Ensure that only one column uses the ACROSS usage code.

FRM1019 The GROUP usage code requires other columns to use aggregate usage codes.

Explanation: When one column uses the GROUP usage code, all of the other columns must use the OMIT, GROUP, BREAK, or AGGREGATION usage codes.

User response: Ensure that all of the other columns use the OMIT, GROUP, BREAK, or AGGREGATION usage codes.

FRM1020 The GROUP usage code requires all other columns to use a non-blank usage code.

Explanation: When a column uses the GROUP usage code, all of the other columns must use the OMIT, GROUP, BREAK, or AGGREGATION usage codes.

User response: Ensure that all of the other columns use the OMIT, GROUP, BREAK, or AGGREGATION usage codes.

FRM1021 The aggregate usage code can only be used when the column has a numeric data type.

Explanation: The SUM, CSUM, AVERAGE, STDEV, PCT, TPCT, CPCT, and TCPCT usage codes can only be used with numeric columns.

User response: Specify a different usage code than SUM, CSUM, AVERAGE, STDEV, PCT, TPCT, CPCT, or TCPCT for non-numeric columns.

FRM1022 There is no calculation expression with ID [0].

Explanation: In the Calculations page of the form designer, the calculation usage code ID must be specified.

User response: Specify a calculation ID for the calculation usage code.

FRM1023 "[0]" is not a valid edit code for a calculation.

Explanation: The specified edit code is not a valid calculation edit code.

User response: Specify another calculation edit code.

FRM1024 Invalid edit code.

Explanation: The specified edit code is invalid.

User response: Check the reference section of the User help to find a valid edit code.

FRM1025 Invalid string "[0]" is encountered. Parsing failed.

Explanation: The form is corrupted or needs to be opened with another locale because a string that represents a number value cannot be parsed in the current locale.

User response: Try to open and save the form with the same version of DataQuant that was used to create the form or try to change the locale and reopen.

FRM1026 The variable "[0]" is invalid in this context.

Explanation: The specified variable name is invalid when used in the expression.

User response: Specify a valid variable name for the expression.
FRM1027  There is no column [0].

Explanation: The expression contains a variable that refers to a column that does not exist.

User response: Specify a valid column number in the expression variable.

FRM1028  The suffix '_B' is invalid in this context.

Explanation: The suffix '_B' can not be used in predefined variables, and as such is invalid for the current variable.

User response: Remove the suffix '_B' from all predefined variables.

FRM1029  Unable to open linked data source.

Explanation: The form attempted to open a linked data source and an error occurred during this process.

User response: Specify a valid data source object for the form.

FRM1030  An error occurred while registering user edit code.


User response: Review the FormCustomEditCode implementation documentation and check for errors.

FRM1031  The user-defined edit routine class could not be found. [0]

Explanation: The user implementation class FormCustomEditCode can not be loaded by the Classloader.

User response: Check the className.jar CLASSPATH.

FRM1032  An error occurred while initializing user-defined edit routine class. [0]

Explanation: An error occurred.

User response: See additional information in the wrapped exception.

GERR00001  Generic driver error has occurred. [0], [1], [2], [3]

Explanation: An internal error in the driver code was detected.

User response: Send an error report to the developers.

GERR00002  Generic driver error has occurred. [0], [1], [2]

Explanation: An internal error in the driver code was detected.

User response: Send an error report to the developers.

GERR00003  Generic driver error has occurred. [0], [1], [2], [3]

Explanation: An internal error in the driver code was detected.

User response: Send an error report to the developers.

GERR00004  Generic driver error has occurred. [0], [1], [2], [3], [4]

Explanation: An internal error in the driver code was detected.

User response: Send an error report to the developers.

GERR00005  Generic driver error has occurred. [0], [1], [2], [3], [4], [5]

Explanation: An internal error in the driver code was detected.

User response: Send an error report to the developers.

GERR00006  Generic driver error has occurred. [0], [1], [2], [3], [4], [5], [6]

Explanation: An internal error in the driver code was detected.

User response: Send an error report to the developers.

GERR00007  Generic driver error has occurred. [0], [1], [2], [3], [4], [5], [6], [7]

Explanation: An internal error in the driver code was detected.

User response: Send an error report to the developers.

GERR00008  Generic driver error has occurred. [0], [1], [2], [3], [4], [5], [6], [7], [8]

Explanation: An internal error in the driver code was detected.

User response: Send an error report to the developers.

IXF01002  The LOB column index is invalid.

Explanation: The file contains an invalid index of a LOB column.

User response: The imported IXF file is corrupted or has an unknown format.

IXF01003  The LOB column length is invalid.

Explanation: The IXF file contains a LOB column that has an invalid length.
User response: The imported IXF file is corrupted or has an unknown format.

IXF01004  The end of the file was encountered prematurely.
Explanation: A premature end of file was encountered while importing data from the IXF file.
User response: The imported IXF file is corrupted or has an unknown format.

IXF01005  The length of record [0] is invalid.
Explanation: A corrupted record was encountered while importing data from the IXF file.
User response: The imported IXF file is corrupted or has an unknown format.

IXF01006  An invalid record length was encountered in record [0].
Explanation: An invalid record length was encountered while importing data from the IXF file.
User response: The imported IXF file is corrupted or has an unknown format.

IXF01007  An invalid name length was encountered in record [0].
Explanation: An invalid column name length was encountered while importing data from the IXF file.
User response: The imported IXF file is corrupted or has an unknown format.

IXF01008  An unrecognized record type was encountered in record [0].
Explanation: An unknown record type was encountered while importing data from the IXF file.
User response: The imported IXF file is corrupted or has an unknown format.

IXF01009  An invalid value was encountered in record [0].
Explanation: An unexpected data value was encountered while importing data from the IXF file.
User response: The imported IXF file is corrupted or has an unknown format.

IXF01010  An unsupported data type was encountered in record [0].
Explanation: An unsupported data type was encountered while importing data from the IXF file.
User response: The imported IXF file is corrupted or has an unknown format.

IXF01011  An unsupported CCSID was encountered in record [0].
Explanation: The IXF file contains character data which is encoded using a CCSID that is not supported by DataQuant.
User response: Create an IXF file using another CCSID to import to DataQuant.

IXF01012  The column length is invalid.
Explanation: An invalid column name was encountered while importing data from the IXF file.
User response: The imported IXF file is corrupted or has an unknown format.

IXF01013  The data record ID in record [0] is negative.
Explanation: An invalid data identifier was encountered while importing data from the IXF file.
User response: The imported IXF file is corrupted or has an unknown format.

IXF01014  The position for a column is negative in record [0].
Explanation: An invalid column position was encountered while importing data from the IXF file.
User response: The imported IXF file is corrupted or has an unknown format.

IXF01015  The "IXF" identifier was not found in record [0].
Explanation: The imported IXF file has no "IXF" signature, which is required at the beginning of its contents.
User response: The imported IXF file is corrupted or has unknown format.

IXF01016  The version of the IXF file is "(0)", which is not supported.
Explanation: The imported IXF file version is not supported by DataQuant.
User response: Check to see if the IXF file is version 0, 1 or 2. All others are not supported.

IXF01017  An invalid record count was encountered in record [0].
Explanation: An invalid column count was encountered while importing data from the IXF file.
User response: The imported IXF file is corrupted or has an unknown format.
IXF01018 • MAIL1000

IXF01018  IXF file contents are corrupted.
Explanation: The imported IXF file is corrupted.
User response: Check that you are specifying an IXF file to import.

IXF01019  Data format is not supported in host IXF files.
Explanation: An unsupported column type was encountered while importing data from the IXF file.
User response: The imported IXF file is corrupted or has an unknown format.

JOB1000  {0}
Explanation:
User response:

JOB1001  Unknown JDF file format.
Explanation: An incorrect job file was selected while importing a job file in the old format (.jdf) to the new format (.jdfx).
User response: Select a correct job file in the old format to import.

JOB1002  Unexpected end of file.
Explanation: A corrupted job file was selected while importing a job file in the old format (.jdf) to the new format (.jdfx).
User response: Resave the job file in QMF version 8 and retry to import the file.

LIC1000  License field "[0]" has invalid date format: [1].
Explanation: The format of the date at the specified field is incorrect.
User response: Ensure that the license file contains a date in dd/mm/yyyy format.

LIC1001  Required field "[0]" missed.
Explanation: No value was specified for the indicated field.
User response: Ensure that a value is specified for the indicated field.

LIC1002  Field "[0]" has invalid value "[1]". Valid value is [2].
Explanation: An incorrect value was specified for the indicated field.
User response: Ensure that a valid license file is being used.

LIC1003  License file could not be found [0].
Explanation: The license file could not be found.
User response: Use the Manage License dialog to add the license to the product.

LIC1004  Unexpected file error: [0].
Explanation: A file system error occurred while loading the license file.
User response: Send an error report to the developers.

LIC1006  License signature verification failed.
Explanation: The license file digital signature verification failed.
User response: Provide a valid license file.

LIC1007  License has expired.
Explanation: The trial period of the license expired.
User response: Buy a full license.

LIC1008  Unexpected error.
Explanation: An unexpected error occurred while verifying the license.
User response: Ensure that the correct license file is being used.

LIC1009  The license is incorrect.
Explanation: The license verification failed because of an invalid license file.
User response: Use the original license file.

LIC1010  License file not found.
Explanation: The license (any *.lic file) was not found the user's home directory or in application's directory.
User response: Use the Manage Licenses dialog to import a valid license.

MAIL1000  {0}
Explanation: The indicated error occurred.
User response: Resolve the error and try again.
MAIL1001  SMTP server must be specified.
Explanation: The email server that will send the message containing the object has not been specified in the procedure command.
User response: Specify the email server that will send the message in the MAIL TO procedure command. Check the MAIL TO procedure command topic in the user help for more information.

MAIL1002  Unknown type of attachment specified.
Explanation: The file specified in the MAIL TO command is of an unknown file format, or was entered incorrectly.
User response: Check the ATTACHMENT parameter of the MAIL TO command for errors or specify a different attachment. Check the MAIL TO procedure command topic in the user help for more information.

NLS00001  Generic error #0 has occurred in module (1).
Explanation: The indicated error occurred.
User response: Send an error report to the developers.

NLS00002  Generic error #0 has occurred in module (1) (2).
Explanation: The indicated error occurred.
User response: Send an error report to the developers.

NLS00003  Generic error #0 has occurred in module (1) (2; 3).
Explanation: The indicated error occurred.
User response: Send an error report to the developers.

NLS00004  Generic error #0 has occurred in module (1) (2; 3; 4).
Explanation: The indicated error occurred.
User response: Send an error report to the developers.

NLS00005  Generic error #0 has occurred in module (1) (2; 3; 4; 5).
Explanation: The indicated error occurred.
User response: Send an error report to the developers.

NLS00006  Generic error #0 has occurred in module (1) (2; 3; 4; 5; 6).
Explanation: The indicated error occurred.
User response: Send an error report to the developers.

NLS00007  Generic error #0 has occurred in module (1) (2; 3; 4; 5; 6; 7).
Explanation: The indicated error occurred.
User response: Send an error report to the developers.

NLS00008  Generic error #0 has occurred in module (1) (2; 3; 4; 5; 6; 7; 8).
Explanation: The indicated error occurred.
User response: Send an error report to the developers.

NLS00009  Generic error #0 has occurred in module (1) (2; 3; 4; 5; 6; 7; 8; 9).
Explanation: The indicated error occurred.
User response: Send an error report to the developers.

OLCQ1000  {0}
Explanation: This is a wrapped message.
User response: Resolve the indicated error and try again.

OLCQ1001  Query was cancelled.
Explanation: The current query was cancelled.
User response: Rerun the query.

OLCQ1002  You must select at least one value for each level.
Explanation: At least one value must be selected in every level of a slicer control.
User response: Select a value and try again.

OLCQ1005  Cannot to find needed data source.
Explanation: A nonexistent data source was specified for the OLAP query.
User response: Specify a valid data source for the OLAP query.

OLP1000  {0}
Explanation: This is a wrapped message.
User response: Resolve the indicated error and try again.

OLP1002  The database does not support OLAP feature.
Explanation: The specified database does not support OLAP functionality.
User response: Specify a database with OLAP functionality.

OLP1003 Unable to locate cube measure {0}.
Explanation: The indicated cube measure could not be found on the database.
User response: Select a different database to search for the cube measure.

OLP1004 Unable to locate cube dimension {0}.
Explanation: The indicated cube dimension could not be found on the database.
User response: Select a different database to search for the cube dimension.

OLP1005 Unable to locate {0}.
Explanation: The indicated OLAP cube could not be found on database.
User response: Select a different database to search for the OLAP cube.

OLP1006 Cannot retrieve values for {0}.
Explanation: An error occurred while reading the dimension information.
User response: Ensure that the indicated dimension exists.

OLP1007 Measure or dimension cannot be added because it is not related to any table currently used in the layout.
Explanation: An error occurred while building the SQL query to retrieve OLAP data from CubeViews; the table that stores information about the selected measure or dimension could not be found.
User response: Check the integrity of the CubeView metadata.

OLP1008 There is not enough information to link specified measure or dimension: {0}.
Explanation: An error occurred while building the SQL query to retrieve OLAP data from CubeViews; the tables that store information about the selected measures or dimensions could not be joined.
User response: Check the integrity of the CubeView metadata.

OLP1009 Cannot connect to database: {0}.
Explanation: Can not connect to the database.
User response: Check if connection to a database is available or ask your administrator for more information.

OLP1010 Cube definition is not suitable for layout of the current query.
Explanation: The cube structure as it is defined in the OLAP query does not match the current cube structure.
User response: Check to make sure that the cube structure that is defined in the OLAP query matches the current cube structure.

OLQ1000 {0}
Explanation: This is a wrapped message.
User response: Resolve the indicated error and try again.

OLQ1001 Internal Error.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

OLQ1002 Query was cancelled.
Explanation: The current query was cancelled.
User response: Rerun the query.

OLQ1003 You must select at least one value for each level.
Explanation: At least one value must be selected in a slicer control.
User response: Select a value and try again.

OLQ1004 Invalid type for saving the query.
Explanation: An invalid object type was selected in which to save the query. OLAP queries can only be saved in the *.oq file format.
User response: Save the OLAP query to a file with the *.oq file extension.

OLQ1005 No data source was specified for the query.
Explanation: A nonexistent data source was specified for the OLAP query.
User response: Specify a valid data source for the OLAP query.
Only one reference to dimension hierarchy {0} can appear in the slicer.

**Explanation:** MDX DB2 DWE supports only one dimension per slicer.

**User response:** Select only one dimension.

---

Encountered "{0}" at column {1}. \r\nWas expecting one of: {2}.

**Explanation:** An invalid character was encountered at the indicated column.

**User response:** Replace the invalid character with one of the indicated characters.

---

Expression needs a calculator to get a value.

**Explanation:** The expression does not return a value.

**User response:** No action is required.

---

Parse exception occurred.

**Explanation:** An error occurred while parsing the expression.

**User response:** Check the expression syntax for errors and try again.

---

An unexpected error occurred while parsing the command.

**Explanation:** An internal error occurred.

**User response:** Collect support data by selecting Help > Collect Support Data and forward it to customer support.

---

An error occurred while parsing the command: unexpected token "[0]".

**Explanation:** An unexpected token was encountered while parsing the procedure.

**User response:** Check the procedure's text for syntax errors (using the documentation or Content Assist via Ctrl+Space)

---

The parameter "[0]" is invalid in this context.

**Explanation:** An unsupported parameter token was encountered while parsing the procedure.

**User response:** Check the procedure's text for syntax errors (using the documentation or Content Assist via Ctrl+Space)

---

The parameter "[0]" cannot be specified twice.

**Explanation:** The indicated parameter was specified twice in the same procedure command.

**User response:** Only specify one instance of the indicated parameter.

---

"[1]" is not a valid value for parameter "[0]".

**Explanation:** The indicated value is not valid for the indicated parameter.

**User response:** Specify a valid value for the indicated parameter. See the Procedure Commands section of the user help for more information on procedure commands and their parameters.

---

You cannot set variable "[0]".

**Explanation:** The indicated variable can not be set manually.

**User response:** Specify a valid variable and try again.

---

An error occurred while parsing the command: unexpected end of line.

**Explanation:** An unexpected end of line was encountered while parsing the procedure.

**User response:** Check the procedure's text for syntax errors (using the documentation or Content Assist via Ctrl+Space)

---

An error occurred while parsing the command: unrecognized command "[0]".

**Explanation:** The indicated command is not valid.

**User response:** Check the procedure text for errors or specify a different command.

---

Appendix C. Messages  501
PRC1010  •  PRC2007

PRC1010  Parsing error at lines {0}-{1} (command #{2}). [3]
Explanation:  a syntax error was encountered while parsing procedure.
User response:  Check the procedure’s text for syntax errors (using the documentation or Content Assist via Ctrl+Space)

PRC1011  An error occurred while parsing the command: invalid object name "]{0}".
Explanation:  The indicated object name is invalid.
User response:  Check the procedure text for errors and specify a valid object name. See the Procedure Commands section of the user help for more information.

PRC1012  You must specify object type or object name.
Explanation:  A required object type or name was not specified.
User response:  Specify a valid object type or name where needed in the procedure command text. See the Procedure Commands section of the user help for more information.

PRC1013  "]{0}" is an invalid variable name.
Explanation:  The indicated variable name is invalid.
User response:  Check the variable name for syntax errors or specify another valid variable. Check the Procedure Commands section of the user help for more information.

PRC1014  Close bracket reached before end.
Explanation:  An unexpected close bracket was encountered while parsing the procedure.
User response:  Check the procedure’s text for syntax errors (using the documentation or Content Assist via Ctrl+Space)

PRC2001  An error occurred while retrieving {0}. [1]
Explanation:  The indicated error occurred while retrieving the indicated object.
User response:  Resolve the error and rerun the procedure.

PRC2002  {0} does not exist.
Explanation:  The indicated object does not exist.
User response:  Specify a valid object in the procedure command.

PRC2003  {0} exists but is of the wrong type.
Explanation:  The indicated object exists in an open window of the editor, but the object type that was specified in the procedure command does not match the type of the object.
User response:  Make sure the object type that is specified in the procedure command matches the type of the object in the open window of the editor.

PRC2004  {0} does not exist or is of the wrong type.
Explanation:  There is no object of the specified object type open in an open window of the editor, or the object type is incorrect.
User response:  Make sure that the object type that is specified in the procedure command matches the type of the object that is in the open window.

PRC2005  The procedure issued a command against a current object, but no current object of the correct type exists.
Explanation:  The procedure does not specify a specific object name, only an object type. In this instance, if an object of this type is open in another window of the editor, the procedure will run correctly. Since an object of the type specified is not open in another window, the procedure failed to run.
User response:  Make sure an object of the specified type is open in another window, or specify a specific object name.

PRC2006  {0}
Explanation:  This is a wrapped message.
User response:  Resolve the indicated error and try again.

PRC2007  The CONVERT command can only be issued against queries; it cannot be applied to {0}.
Explanation:  The indicated object can not be converted using the CONVERT command.
User response:  Specify a query to be converted using the CONVERT command. For more information, see the CONVERT topic in the Procedure Commands section of the user help.
PRC2008 The operation was cancelled.
Explanation: The operation was cancelled by the user.
User response: Rerun the operation.

PRC2011 Saving in [0] format is not supported.
Explanation: The selected format is not supported when exporting an object. Reports may only be exported in TEXT, HTML, and PDF formats.
User response: Change the export data format in the procedure command.

PRC2017 The maximum nesting depth of procedures was exceeded.
Explanation: More than 50 nested procedure levels are not support.
User response: Redesign the procedure to decrease its depth.

PRC2018 The specified query did not return a result set.
Explanation: The SAVE DATA command was issued against a query that does not contain a result set.
User response: Ensure that the query you are trying to save contains a result set.

PRC2020 An error occurred while saving the object at the server. [0]
Explanation: The indicated error occurred while the object was being saved at the server.
User response: Resolve the indicated error and rerun the procedure.

PRC2022 The operation cannot be performed because no data is available.
Explanation: The EXPORT DATA command was issued against an object that does not contain data.
User response: Ensure that the object that you are trying to export contains data.

PRC2025 The procedure was terminated because it recursively invokes itself.
Explanation: The procedure recursively invoked itself. Recursion is not supported in procedures.
User response: Rewrite the procedure and remove the recursion.

PRC2026 An error occurred while running the procedure. [0]
Explanation: The indicated error occurred while the procedure was being run.
User response: Resolve the indicated error and rerun the procedure.

PRC2028 An error occurred while importing the [0] file.
Explanation: An error occurred while the file was being imported.
User response: Ensure that object that is being imported is of a supported format.

PRC2029 Could not connect to a server [1]: [0]
Explanation: The data source alias that was specified in the CONNECT TO command is undefined.
User response: Ensure that the command refers to a valid data source in the repository.

PRC2030 Specified object [0] for import has incompatible type.
Explanation: The IMPORT or DISPLAY command was issued against an object with an unsupported type.
User response: Ensure that the object to be imported or displayed is of a valid type.

PRC2031 Specified command, or some of its parameters unsupported.
Explanation: The procedure contains commands or parameters that are not supported.
User response: Check the procedure's text for syntax errors (using the documentation or Content Assist via Ctrl+Space)

PRC2032 Cannot save non-QMF object into QMF Catalog.
Explanation: The specified object is a non-QMF object and can not be saved into the QMF Catalog.
User response: Specify a different location in which the save the object or specify a different object to be saved.

PRC2033 You cannot set the value of [0].
Explanation: The indicated parameter's value can not be manually set with the SET GLOBAL command.
User response: Values for some DSQ variables cannot be set manually (for example DSQAO_CONNECT_ID, DSQAO_NUM_FETCHED, etc). Rewrite the procedure to exclude these variable values.
PRC2035  The printer name "[0]" is invalid.
Explanation:  The indicated printer name is invalid.
User response:  Check the spelling of the printer name, or specify a different printer to use for the PRINT command.

PRC2036  You are not authorized to use the table editor at this data source.
Explanation:  The Enable Table Editing resource limit for the data source is turned off; you cannot open Table Editor of the data source.
User response:  Ask your administrator to enable this resource limit.

PRC2037  You are not authorized to edit tables at this data source.
Explanation:  The Enable Table Editing resource limit for the data source is turned off; you cannot edit values in a query's grid cells.
User response:  Ask your administrator to enable this resource limit.

PROC1000  REXX procedure references cannot be determined.
Explanation:  References for the procedure cannot be built for REXX procedures.
User response:  You can view references at the Repository or Workspaces views for generic procedures only.

PROC1001  Some substitution variables are not defined.
Explanation:  Some of the substitution variables of the procedure are not defined.
User response:  Ensure that all of the substitution variables are defined for the procedure, or run this procedure without the /batch parameter in the command line. In this case DataQuant will ask for all of the necessary variables values.

QEL00001  You are not allowed to run queries returning LOB data.
Explanation:  You are not allowed to run queries that contain large object data.
User response:  Check your LOB Options resource limits by selecting Resource Limits from the View menu to see your specific allowances or ask your administrator for more information.

QEL00002  The Maximum Rows to Fetch limit has been exceeded. [0] rows were retrieved so far.
Explanation:  The Maximum Rows to Fetch limit has been reached. The indicated number of rows have been retrieved so far.
User response:  Check your Maximum Rows to Fetch resource limit by selecting Resource Limits from the View menu to see your specific allowances or ask your administrator for more information.

QEL00003  The Maximum Bytes to Fetch limit has been exceeded. [0] bytes were retrieved so far.
Explanation:  The Maximum Bytes to Fetch limit has been reached. The indicated number of bytes have been retrieved so far.
User response:  Check your Maximum Bytes to Fetch resource limit by selecting Resource Limits from the View menu to see your specific allowances or ask your administrator for more information.

QEL00004  The length of a LOB column exceeds the maximum allowable length.
Explanation:  The length of a LOB column in the query results exceeds the maximum allowable length.
User response:  Check your Maximum LOB column size by selecting Resource Limits from the View menu to see your specific allowances or ask your administrator for more information.

QEL00005  You are not allowed to use the "[0]" SQL verb.
Explanation:  You are not allowed to use the indicated SQL verb.
User response:  Check your SQL Verbs limits by selecting Resource Limits from the View menu to see your specific allowances or ask your administrator for more information.

QEL00006  You are not allowed to save data to database.
Explanation:  You are not allowed to save data to the database.
User response:  Check your Save Data resource limits by selecting Resource Limits from the View menu to see your specific allowances or ask your administrator for more information.
<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>QEN00001</td>
<td>Query was cancelled.</td>
</tr>
<tr>
<td></td>
<td>Explanation: The active query was cancelled.</td>
</tr>
<tr>
<td></td>
<td>User response: Rerun the query.</td>
</tr>
<tr>
<td>QEN00003</td>
<td>Unexpected token: {0}</td>
</tr>
<tr>
<td></td>
<td>Explanation: An unexpected token was encountered while parsing the query that calls the stored procedure.</td>
</tr>
<tr>
<td></td>
<td>User response: Check the query's syntax for errors.</td>
</tr>
<tr>
<td>QEN00004</td>
<td>Unexpected token: {0} is expected but {1} was found.</td>
</tr>
<tr>
<td></td>
<td>Explanation: An unexpected closed bracket was encountered while parsing the query that calls the stored procedure.</td>
</tr>
<tr>
<td></td>
<td>User response: Check the query's syntax for errors.</td>
</tr>
<tr>
<td>QEN00005</td>
<td>Unexpected end of statement.</td>
</tr>
<tr>
<td></td>
<td>Explanation: An unexpected end of statement was encountered while parsing the query that calls the stored procedure.</td>
</tr>
<tr>
<td></td>
<td>User response: Check the query's syntax for errors.</td>
</tr>
<tr>
<td>QEN00006</td>
<td>Expression {0} is not a stored procedure.</td>
</tr>
<tr>
<td></td>
<td>Explanation: An internal error occurred.</td>
</tr>
<tr>
<td></td>
<td>User response: Collect support data by selecting Help &gt; Collect Support Data and forward it to customer support.</td>
</tr>
<tr>
<td>QEN00007</td>
<td>End of statement is expected but {0} was found.</td>
</tr>
<tr>
<td></td>
<td>Explanation: Tokens were encountered after the end of statement while parsing the query that calls the stored procedure.</td>
</tr>
<tr>
<td></td>
<td>User response: Check the query's syntax for errors.</td>
</tr>
<tr>
<td>QEN00008</td>
<td>Unsupported parameter data type: {0}</td>
</tr>
<tr>
<td></td>
<td>Explanation: An unsupported parameter was encountered while parsing the query that calls the stored procedure.</td>
</tr>
<tr>
<td></td>
<td>User response: Check the query's syntax for errors.</td>
</tr>
<tr>
<td>QEN00009</td>
<td>{0} is invalid value for this type: {1}</td>
</tr>
<tr>
<td></td>
<td>Explanation: When prompted for a value for a stored procedure parameter, the user entered a value that is not valid for that parameter's type.</td>
</tr>
<tr>
<td></td>
<td>User response: Specify a valid value for the parameter.</td>
</tr>
<tr>
<td>QMF1004</td>
<td>You are not authorized to print objects from &quot;{0}&quot; data source.</td>
</tr>
<tr>
<td></td>
<td>Explanation: The indicated data source name has a restriction on printing for the current user.</td>
</tr>
<tr>
<td></td>
<td>User response: Contact your administrator to check your specific permissions for printing.</td>
</tr>
<tr>
<td>QMF1005</td>
<td>QMF Catalog table structure is obsolete. Catalog tables must be updated.</td>
</tr>
<tr>
<td></td>
<td>Explanation: The indicated an error occurred because the QMF Catalog has an obsolete structure.</td>
</tr>
<tr>
<td></td>
<td>User response: Contact your administrator to upgrade the QMF Catalog.</td>
</tr>
<tr>
<td>QMF1006</td>
<td>You are not authorized to save data to files from &quot;{0}&quot; data source.</td>
</tr>
<tr>
<td></td>
<td>Explanation: The indicated data source name has a restriction on saving data for the current user.</td>
</tr>
<tr>
<td></td>
<td>User response: Contact your administrator to check your specific permissions for saving data.</td>
</tr>
<tr>
<td>QRL00001</td>
<td>You are not authorized to access this data source in user interface mode.</td>
</tr>
<tr>
<td></td>
<td>Explanation: You are not authorized to access this data source in user interface mode.</td>
</tr>
<tr>
<td></td>
<td>User response: Check your Options limits by selecting Resource Limits from the View menu to see your specific allowances or ask your administrator for more information.</td>
</tr>
<tr>
<td>QRL00002</td>
<td>You are not authorized to access this data source in automation mode.</td>
</tr>
<tr>
<td></td>
<td>Explanation: You are not authorized to access this data source in automation mode.</td>
</tr>
<tr>
<td></td>
<td>User response: Check your Options limits by selecting Resource Limits from the View menu to see your specific allowances or ask your administrator for more information.</td>
</tr>
<tr>
<td>QRL00003</td>
<td>You are not authorized to run unsaved queries.</td>
</tr>
<tr>
<td></td>
<td>Explanation: You are not authorized to run unsaved queries.</td>
</tr>
<tr>
<td></td>
<td>User response: Check your Options limits by selecting Resource Limits from the View menu to see your specific allowances or ask your administrator for more information.</td>
</tr>
</tbody>
</table>
You are not authorized to save data from this data source to a file.

**Explanation:** You are not authorized to save data from this data source to a file.

**User response:** Check your Save Data limits by selecting Resource Limits from the View menu to see your specific allowances or ask your administrator for more information.

You are not authorized to save data from this data source to the database.

**Explanation:** You are not authorized to save data from this data source to the database.

**User response:** Check your Save Data limits by selecting Resource Limits from the View menu to see your specific allowances or ask your administrator for more information.

You are not authorized to retrieve LOB data.

**Explanation:** You are not authorized to retrieve LOB data.

**User response:** Check your LOB Options limits by selecting Resource Limits from the View menu to see your specific allowances or ask your administrator for more information.

No resource group for user [0] was found.

**Explanation:** No resource group for the indicated user was found.

**User response:** Specify a valid resource group for the user.

The operation cannot be performed because the processing of LOB data is disabled.

**Explanation:** The operation cannot be performed because the processing of LOB data is disabled.

**User response:** Check you LOB Options limits by selecting Resource Limits from the View menu to see your specific allowances or ask your administrator for more information.

This is a wrapped message.

**Explanation:** This is a wrapped message.

**User response:** Resolve the indicated error and try again.

Query was cancelled.

**Explanation:** The query was cancelled.

**User response:** Rerun the query.

An invalid header record was encountered.

**Explanation:** The query was loaded with errors.

**User response:** Try to reload the query; otherwise the query is corrupted.

No data source was specified for the query.

**Explanation:** No data source was specified for the query.

**User response:** Specify a data source for the query.

An unexpected "R" record was encountered: [0].

**Explanation:** An error was encountered while loading a type R record from the prompted query file during storage of info in the prompted query structure.

**User response:** Collect support data by selecting Help > Collect Support Data and forward it to customer support.

An unexpected "T" record was encountered: [0].

**Explanation:** An error was encountered while loading a type T record from the prompted query file during storage of info in the prompted query structure.

**User response:** Collect support data by selecting Help > Collect Support Data and forward it to customer support.

An invalid "T" record was encountered: [0].

**Explanation:** An error was encountered while loading a type T record from the prompted query file during storage of info in the prompted query structure.

**User response:** Collect support data by selecting Help > Collect Support Data and forward it to customer support.

A "T" record with invalid column information was encountered: [0].

**Explanation:** An error was encountered while loading a type T record from the prompted query file during storage of info in the prompted query structure.

**User response:** Collect support data by selecting Help.
> Collect Support Data and forward it to customer support.

---

**QRY1008**  
**An unrecognized table type was encountered:** {0}.

**Explanation:**  
An error was encountered while loading a record from the prompted query file during storage of info in the prompted query structure.

**User response:**  
Collect support data by selecting Help > Collect Support Data and forward it to customer support.

---

**QRY1009**  
**An invalid value in an "R" record for table 1110 was encountered:** {0}.

**Explanation:**  
An error was encountered while loading a type R record from the prompted query file during storage of info in the prompted query structure.

**User response:**  
Collect support data by selecting Help > Collect Support Data and forward it to customer support.

---

**QRY1010**  
**An invalid value in an "R" record for table 1150 was encountered:** {0}.

**Explanation:**  
An error was encountered while loading a type R record from the prompted query file during storage of info in the prompted query structure.

**User response:**  
Collect support data by selecting Help > Collect Support Data and forward it to customer support.

---

**QRY1011**  
**An invalid value in an "R" record for table 1210 was encountered:** {0}.

**Explanation:**  
An error was encountered while loading a type R record from the prompted query file during storage of info in the prompted query structure.

**User response:**  
Collect support data by selecting Help > Collect Support Data and forward it to customer support.

---

**QRY1012**  
**An invalid value in an "R" record for table 1310 was encountered:** {0}.

**Explanation:**  
An error was encountered while loading a type R record from the prompted query file during storage of info in the prompted query structure.

**User response:**  
Collect support data by selecting Help > Collect Support Data and forward it to customer support.

---

**QRY1013**  
**An invalid value in an "R" record for table 1410 was encountered:** {0}.

**Explanation:**  
An error was encountered while loading a type R record from the prompted query file during storage of info in the prompted query structure.

**User response:**  
Collect support data by selecting Help > Collect Support Data and forward it to customer support.

---

**QRY1014**  
**An out of sequence 'R' record for table 1310 was encountered.**

**Explanation:**  
An error was encountered while loading a type R record from the prompted query file during storage of info in the prompted query structure.

**User response:**  
Collect support data by selecting Help > Collect Support Data and forward it to customer support.

---

**QRY1015**  
**An invalid "V" record was encountered:** {0}.

**Explanation:**  
An error was encountered while loading a type V record from the prompted query file during storage of info in the prompted query structure.

**User response:**  
Collect support data by selecting Help > Collect Support Data and forward it to customer support.

---

**QRY1016**  
**There must be less than 16 tables in Prompted Query.**

**Explanation:**  
There must be less than sixteen tables called from the Prompted Query editor.

**User response:**  
Ensure that there are less than sixteen tables called from the Prompted Query editor and rerun the query.

---

**QRY1017**  
**The same correlation ID was encountered for two or more tables.**

**Explanation:**  
The PqTable object for the specified correlation ID already exists.

**User response:**  
Select another table for the Prompted Query editor.

---

**QRY1018**  
**An invalid table correlation ID was encountered.**

**Explanation:**  
An error was encountered while retrieving a valid correlation ID.

**User response:**  
Try to add the query to the Prompted Query editor again, or collect support data by selecting Help > Collect Support Data and forward it to customer support.
A column on which to join tables is in more than one of the tables in the query.

**Explanation:** A join column can only exist in one table when multiple tables are joined in a single query.

**User response:** Ensure that any join columns in the query exist in only one table.

A column on which to join tables is not in any of the tables in the query.

**Explanation:** A specified join column is not present in any of the tables in the query.

**User response:** Check the join column name for errors, or specify a different join column.

Type of join columns for joined tables mismatch.

**Explanation:** Columns that join two or more tables must all be of the same column type.

**User response:** Ensure that all of the columns that make up a single join are of the same column type.

The expression contains a syntax error: {0}

**Explanation:** The indicated expression contains a syntax error.

**User response:** Correct the syntax error and rerun the query.

An invalid connector for a row condition was encountered.

**Explanation:** An invalid connector for a row condition was encountered.

**User response:** Check all row conditions for errors and rerun the query.

An invalid expression type in a row condition was encountered.

**Explanation:** An invalid expression type in a row condition was encountered.

**User response:** Check all expression types in the row conditions for errors and rerun the query.

An invalid operator in a row condition was encountered.

**Explanation:** An invalid operator in a row condition was encountered.

**User response:** Check all operators in the row conditions for errors and rerun the query.

The selected relational operator is not valid for numeric data.

**Explanation:** An invalid operator in a row condition was encountered for numeric data.

**User response:** Check all of the operators in the row conditions for errors and rerun the query.

An invalid sort direction was encountered.

**Explanation:** An invalid sort direction was encountered. Valid directions are Ascending or Descending.

**User response:** Check all instances where sort directions are applied and change them to either ascending or descending.

The prompted query format is an earlier version, and contains join conditions that cannot be migrated to the current version. These join conditions have been deleted.

**Explanation:** A join condition can only appear with the higher of the two tables involved in the join. Also, a table cannot be joined with a preceding table if there is a third table between them with no join conditions.

**User response:** Reordered elements of the array of joins to match the current order of the tables.

The table {0} does not exist.

**Explanation:** The indicated table is called in the SQL code, but does not exist in the data source.

**User response:** Check the table name for errors, or enter a different table to be called.

The prompted query cannot be saved because a table name exceeds the maximum length allowed in the prompted query export format.

**Explanation:** The prompted query cannot be saved because a table name exceeds the maximum length allowed in the prompted query export format.

**User response:** Decrease the length of the table name.

The prompted query cannot be saved because a column name used in a join condition exceeds the maximum length allowed in the prompted query export format.

**Explanation:** The prompted query cannot be saved because a column name used in a join condition exceeds the maximum length allowed in the prompted query export format.

**User response:** Decrease the length of the column name.
User response: Decrease the length of the join column name.

QRY1032 The prompted query cannot be saved because a column expression exceeds the maximum length allowed in the prompted query export format.

Explanation: The prompted query cannot be saved because a column expression exceeds the maximum length allowed in the prompted query export format.

User response: Decrease the length of the column expression name.

QRY1033 The prompted query cannot be saved because an expression in a row condition exceeds the maximum length allowed in the prompted query export format.

Explanation: The prompted query cannot be saved because an expression in a row condition exceeds the maximum length allowed in the prompted query export format.

User response: Decrease the length of the row condition.

QRY1034 An invalid record was encountered.

Explanation: An invalid record was encountered.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

QRY1035 Only a SELECT query can be shown in the prompted view.

Explanation: Only a SELECT query can be shown in the Prompted Query editor.

User response: Make sure that the specified query contains a SELECT statement.

QRY1036 Only simple SELECT statements are supported.

Explanation: Only simple SELECT statements are supported.

User response: Ensure that the specified query only contains simple SELECT statements.

QRY1037 Unrecognized table reference.

Explanation: An error occurred while parsing the SQL text to construct the prompted query.

User response: Check the SQL text for errors and try again.

QRY1038 You cannot view a query that has subselects in the prompted view.

Explanation: Subselects are not supported for queries that are viewed in the Prompted Query editor.

User response: Ensure that specified query does not contain subselects if it is to be viewed in the Prompted Query editor.

QRY1039 Table locator references are not supported.

Explanation: An error occurred while parsing the SQL text to construct the prompted query.

User response: Check the SQL text for errors and try again.

QRY1040 Table functions are not supported.

Explanation: An error occurred while parsing the SQL text to construct the prompted query.

User response: Check the SQL text for errors and try again.

QRY1041 Only AND conditions are supported for joins in the prompted view.

Explanation: Only AND conditions are supported for joins in queries that are viewed in the Prompted Query editor.

User response: Ensure that the specified query only contains AND conditions for joins if it is to be viewed in the Prompted Query editor.

QRY1042 Quantified predicates are not supported.

Explanation: An error occurred while parsing the SQL text to construct the prompted query.

User response: Check the SQL text for errors and try again.

QRY1043 'Exists' predicates are not supported.

Explanation: An error occurred while parsing the SQL text to construct the prompted query.

User response: Check the SQL text for errors and try again.

QRY1044 Only simple LIKE statements are supported.

Explanation: Only simple LIKE statements are supported.

User response: Ensure that the specified query only contains simple LIKE statements.
<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
<th>Explanation</th>
<th>User response</th>
</tr>
</thead>
<tbody>
<tr>
<td>QRY1045</td>
<td>Only simple row conditions are supported.</td>
<td>Only simple row conditions are supported.</td>
<td>Ensure that the specified query only contains simple row conditions.</td>
</tr>
<tr>
<td>QRY1046</td>
<td>Expression cannot be empty.</td>
<td>An error occurred while constructing the query.</td>
<td>Check the query's syntax for errors and try again.</td>
</tr>
<tr>
<td>QRY1047</td>
<td>Invalid type for saving the query.</td>
<td>The type that was set for the query is invalid.</td>
<td>Specify a valid type for the query or collect support data by selecting Help &gt; Collect Support Data and forward it to customer support.</td>
</tr>
<tr>
<td>QRY1048</td>
<td>You cannot run an empty query.</td>
<td>The specified query did not contain any SQL code.</td>
<td>Populate the query with at least a SELECT statement and rerun.</td>
</tr>
<tr>
<td>QRY1049</td>
<td>Some substitution variables are not defined.</td>
<td>Some of the substitution variables in the query were not given values when the query was run.</td>
<td>Ensure that all of the substitution variables in the query are defined.</td>
</tr>
<tr>
<td>QRY1050</td>
<td>You cannot view a query that uses row values in the predicate.</td>
<td>An error occurred while parsing the SQL text to construct the prompted query.</td>
<td>Check the SQL text for errors and try again.</td>
</tr>
<tr>
<td>QRY1051</td>
<td>'XMLExists' predicates are not supported.</td>
<td>An error occurred while parsing the SQL text to construct the prompted query.</td>
<td>Check the SQL text for errors and try again.</td>
</tr>
<tr>
<td>QRY1052</td>
<td>An invalid value: {0}.</td>
<td>An error occurred while parsing the SQL text to construct the prompted query.</td>
<td>Check the SQL text for errors and try again.</td>
</tr>
<tr>
<td>QRY1053</td>
<td>Only SELECT statements are supported in compound queries.</td>
<td>Only SELECT statements are supported in compound queries.</td>
<td>Ensure that only SELECT statements are called in the specified compound query.</td>
</tr>
<tr>
<td>QRY1054</td>
<td>Query content is not compatible with Prompted Query format.</td>
<td>The query can not be saved in the prompted format.</td>
<td>Save the query in another format.</td>
</tr>
<tr>
<td>QRY1055</td>
<td>Host variable predicates are not supported.</td>
<td>An error occurred while parsing the SQL text to construct the prompted query.</td>
<td>Check the SQL text for errors and try again.</td>
</tr>
<tr>
<td>RCO0000</td>
<td>Internal error</td>
<td>An internal processor error occurred.</td>
<td>Collect support data by selecting Help &gt; Collect Support Data and forward it to customer support.</td>
</tr>
<tr>
<td>RCO0001</td>
<td>Type mismatch</td>
<td>An unexpected error occurred while processing the *.rco file.</td>
<td>Collect support data by selecting Help &gt; Collect Support Data and forward it to customer support.</td>
</tr>
<tr>
<td>RCO0002</td>
<td>Function {0} is not implemented yet</td>
<td>An unexpected error occurred while processing the *.rco file.</td>
<td>Collect support data by selecting Help &gt; Collect Support Data and forward it to customer support.</td>
</tr>
</tbody>
</table>
RCO0003  Undefined variable {0}
Explanation: An unexpected error occurred while processing the *.rco file.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RCO0004  Missing variable name
Explanation: An unexpected error occurred while processing the *.rco file.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RCO0005  Syntax error in \"{0}\" 
Explanation: An unexpected error occurred while processing the *.rco file.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RCO0006  Unknown function {0}
Explanation: An unexpected error occurred while processing the *.rco file.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RCO0007  Unknown object type {0}
Explanation: An unexpected error occurred while processing the *.rco file.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RCO0008  Wrong parameters in \"{0}\" 
Explanation: An unexpected error occurred while processing the *.rco file.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RCO0009  Database object {0} not found
Explanation: An unexpected error occurred while processing the *.rco file.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RCO0010  Database error {0}
Explanation: An unexpected error occurred while processing the *.rco file.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RCO0011  Unknown database server type {0}
Explanation: An unexpected error occurred while processing the *.rco file.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

REP01000  {0}
Explanation: The indicated error occurred while accessing the repository.
User response: Resolve the indicated error and try again.

REP01001  An input/output error occurred: {0}
Explanation: The indicated error occurred in the repository.
User response: Resolve the indicated error and try again.

REP01002  Character encoding is not supported: {0}
Explanation: The indicated encoding error occurred while the data was being processed.
User response: Ensure that UTF-8 encoding is used in the repository storage database.

REP01003  Internal error has occurred: {0}
Explanation: The indicated error occurred.
User response: Resolve the indicated error and try again.

REP01004  The following error has occurred while accessing database: {0}
Explanation: The indicated error occurred while the database was being accessed.
User response: Resolve the indicated error and retry to connect to the database.

REP01005  Operation is not supported.
Explanation: The specified operation is not supported by the repository.
User response: Specify a different operation and try again.

REP01006  Object with such identifier not found.
Explanation: An error occurred while searching for the object.
User response: Ensure that the specified object identifier is correct.

REP01007  Object or file with URL "{0}" not found.
Explanation: There are no objects or files with the indicated URL.
User response: Check the URL for errors or specify a different URL.

REP01008  Not enough permissions to perform operation.
Explanation: Your user ID does not have permission to perform the specified operation.
User response: Contact your administrator to check your specific permissions or specify a different operation to perform.

REP01009  Operation cancelled due to invalid object identifier: {0}
Explanation: The operation was cancelled because the indicated object identifier is invalid.
User response: Specify a different object identifier and retry the specified operation.

REP01010  Cannot open connection: {0}
Explanation: The indicated error occurred while establishing a connection to the repository database.
User response: Ensure that the repository database is accessible.

REP01011  At least one line of the document is longer than 79 characters, which is the maximum supported length when saving objects in the database.
Explanation: The maximum character length allowed for documents saved to the database is seventy-nine characters. The document contains at least one line that is longer than seventy-nine characters.
User response: Ensure that all of the lines of the document are less than seventy-nine characters and retry to save the document.

REP01012  {0} already exists and has a different type.
Explanation: The indicated object name already exists as a different type of object. Each object in the database must have a different name.
User response: Specify a different name for the object and retry to save the object.

REP01013  {0} does not exist.
Explanation: The indicated object could not be found because it does not exist in the database.
User response: Specify a different object name to search for in the database.

REP01014  {0} already exists.
Explanation: The indicated object could not be saved because it already exists in the database.
User response: Specify a different name for the object and retry to save the object.

REP01015  {0} has unsupported type and cannot be accessed.
Explanation: The indicated object is of an unsupported type.
User response: Ensure that all of the necessary plug-ins are installed and available.

REP01016  Operation is impossible because object is too large.
Explanation: The specified operation could not be performed because the specified object is too large.
User response: Specify a different object or a different operation and try again.

REP01017  Illegal state or illegal operation error.
Explanation: An internal error occurred in the repository.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

REP01018  User is not logged on repository.
Explanation: You can not perform the specified operation, because you are not connected to the repository.
User response: Connect to the repository and try again.
|REP01019| Not implemented.  
**Explanation:** The indicated error occurred while attempting to access a non-implemented functionality. 
**User response:** Ensure that you are using the latest version of the product. |
|---|---|
|REP01020| Unexpected error occurred.  
**Explanation:** An internal error occurred in the repository. 
**User response:** Collect support data by selecting Help > Collect Support Data and forward it to customer support. |
|---|---|
|REP01021| Initialization error occurred "{0}".  
**Explanation:** An error occurred while the repository was being initialized. 
**User response:** Check the repository initialization parameters and try again. |
|---|---|
|REP01022| Invalid initialization parameters were specified "{0}".  
**Explanation:** An error occurred while the repository was being initialized. 
**User response:** Check the repository initialization parameters and try again. |
|---|---|
|REP01027| A repository instance hasn't been initialized.  
**Explanation:** An error occurred while trying to access a non-initialized repository. 
**User response:** Initialize the repository and try again. |
|---|---|
|REP01028| Repository "{0}" could not be found.  
**Explanation:** The indicated repository could not be found. 
**User response:** Check the repository name for errors or specify a different repository and try again. |
|---|---|
|REP01029| Object type {0} cannot be added to this folder.  
**Explanation:** Objects of the indicated type can not be added to the specified folder. 
**User response:** Specify a different folder in which to add the indicated object. |
|---|---|
|REP01030| More authentication information for {0} required.  
**Explanation:** Not enough authentication information was provided for the indicated object. 
**User response:** Provide more authentication information to the indicated object and try again. |
|---|---|
|REP01031| Repository storage table structure is obsolete. Contact repository storage administrator.  
**Explanation:** An error occurred while trying to access a repository that has an obsolete structure. 
**User response:** Contact your administrator to upgrade the repository database structure. |
|---|---|
|REP01032| CCSID information could not be found for current QMF catalog.  
**Explanation:** An error occurred while trying to read CCSID information from the QMF Catalog. 
**User response:** Specify valid CCSID information and try again. |
|---|---|
|REP01034| A repository with name {0} already exists.  
**Explanation:** The repository could not be created, because a repository of the same name already exists. 
**User response:** Specify a different name for the new repository and try again. |
|---|---|
|REP01035| Unknown object type.  
**Explanation:** An error occurred while trying to access an unsupported object. 
**User response:** Ensure that all of the required plug-ins are installed and available. |
|---|---|
|REP01036| Object {0} cannot be added to this folder.  
**Explanation:** The indicated object can not be added to the specified folder. 
**User response:** Specify a different folder in which to add the indicated object. |
|---|---|
|REP01037| Repository could not be found.  
**Explanation:** The specified repository could not be found. 
**User response:** Check the repository name for errors or specify a different repository name and try again. |
REP01038  Undefined or wrong repository identifier
Explanation: An error occurred while trying to initialize a repository with deprecated initialization parameters.
User response: Change the repository connection properties through the Edit Repository Connection dialog and try again.

REP01039  There are no catalogs on the server
Explanation: No QMF catalogs exist on the specified server.
User response: Specify a different server and try again.

REP01042  You are not authorized to use objects with Owner = [0].
Explanation: Your user ID is not authorized to use objects with the indicated owner.
User response: Specify an object that has a different owner or contact your administrator for more information.

REP01043  This data source cannot be accessed by DataQuant
Explanation: The application can not access this data source.
User response: Check your user license and try again.

REP01044  Access to data source "/[0]" blocked due to QMF catalog restriction violation.
Explanation: Access to the indicated data source is blocked due to a restriction imposed by the QMF catalog protection limits.
User response: Specify a different data source to access or contact your administrator for more information.

REP01045  Associated QMF catalog data source could not be found: [0]
Explanation: The indicated QMF catalog data source could not be found.
User response: Check the data source name for errors or specify a different data source and try again.

REP01046  Cannot find connection: [0]
Explanation: The indicated repository connection could not be found.
User response: Specify a different repository connection and try again.

REP01047  Cannot find data source: [0]
Explanation: The indicated data source could not be found.
User response: Specify a different data source and try again.

REP01049  Object name cannot exceed [0] characters.
Explanation: The indicated object name exceeds the maximum character limit.
User response: Shorten the indicated object name and try again.

REP01050  Object path cannot exceed [0] characters.
Explanation: The indicated object path exceeds the character limit.
User response: Shorten the indicated object path and try again.

REP01051  Cannot connect to repository "/"\"/[0]\"" because it is used by another instance of DataQuant. Select different repository.
Explanation: You can not connect to the indicated repository because it is being used by another instance of the application.
User response: Select a different repository connection and try again.

REP01052  Data Source plug-in "/"\"/[0]\"" was not properly initialized.
Explanation: The indicated plug-in was not properly initialized.
User response: Change the plug-in parameters.

RGEN1000  An error has occurred while generating the report: [0]
Explanation: The indicated error occurred while the report was being generated.
User response: Resolve the indicated error and retry to generate the report.

RGEN1001  The action was cancelled.
Explanation: The specified action was cancelled
User response: Rerun the specified action.
RGEN1002 The number of columns in the form does not match the number of columns in the query.
Explanation: The number of columns in the form does not match the number of columns in its associated query.
User response: Edit the number of columns in the form to match the number of columns its associated query.

RGEN1003 Internal error.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RGEN1004 The number of columns in the form does not match the number of columns in the query.
Explanation: The number of columns in the form does not match the number of columns in its associated query.
User response: Edit the number of columns in the form to match the number of columns its associated query.

RLE00002 Cannot retrieve resource limits of type {0}.
Explanation: The resources of the indicated type cannot be retrieved.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RLE00003 An error has occurred while retrieving resource limits from remote site: {0}
Explanation: The indicated error occurred while resource limits were being obtained from a remote site.
User response: Resolve the indicated error and try again.

RLE00004 Unknown resource limits key is used to retrieve limits.
Explanation: An error occurred while retrieving resource limits.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RLE00005 An error has occurred while obtaining resource limits: {0}
Explanation: The indicated error occurred while resource limits were being obtained.
User response: Resolve the indicated error and try again.

RLE00006 Cannot retrieve resource limits because resource limits key content is corrupted. {0}
Explanation: The indicated error occurred while attempting to retrieve resource limits.
User response: Resolve the indicated error and try again.

RLE00007 The following error has occurred while retrieving resource limits: {0}
Explanation: The indicated error occurred while resource limits were being retrieved.
User response: Resolve the error and try again.

RLE00008 Data source cannot be accessed.
Explanation: Your user ID does not have permission to access the data source.
User response: Check your resource limits by selecting Resource Limits from the View menu or contact your administrator for more information.

RLE00009 LOB data saving is denied.
Explanation: Your user ID does not have permission to save objects that contain LOB data.
User response: Check your LOB Options resource limits by selecting Resource Limits from the View menu or contact your administrator for more information.

RLE00010 You are only authorized to run saved queries at this server.
Explanation: You are only allowed to run saved queries while you are connected to the current server.
User response: Check you Save Data resource limits by selecting Resource Limits from the View menu or
contact your administrator for more information.

---

**RLE00011** You are not authorized to save objects at this server.

**Explanation:** You are not authorized to save objects while you are connected to the current server.

**User response:** Check your Save Data resource limits by selecting **Resource Limits** from the **View** menu or contact your administrator for more information.

---

**RLE00012** You are not authorized to access data source [0].

**Explanation:** Your user ID is not authorized to access the indicated data source.

**User response:** Try to connect to another data source, or contact your administrator for information regarding specific allowances.

---

**RLE00013** You are not authorized to access data source [0].

**Explanation:** Your user ID is not authorized to access the indicated data source.

**User response:** Try to connect to another data source, or contact your administrator for information regarding specific allowances.

---

**RLW00001** [0] rows were retrieved. The Maximum Rows to Fetch warning limit has been exceeded.

**Explanation:** The indicated number of rows were retrieved and the warning limit for the **Maximum Rows to Fetch** limit has been reached.

**User response:** Click OK to continue running the query. Click **Abort** to abort the query. Click **Ignore** to ignore the warning.

---

**RLW00002** [0] bytes were retrieved. The Maximum Bytes to Fetch warning limit has been exceeded.

**Explanation:** The indicated number of bytes were retrieved and the warning limit for the **Maximum Bytes to Fetch** limit has been reached.

**User response:** Click OK to continue running the query. Click **Abort** to abort the query. Click **Ignore** to ignore the warning.

---

**RLW00003** There has been no response from [0],[1] for [2] seconds.

**Explanation:** The query was cancelled because the Server Response Timeout limit has been reached. There was no response from the indicated server after a wait time of the indicated length.

**User response:** Check your Server Response Timeouts resource limit by selecting **Resource Limits** from the **View** menu or try to connect to the server during a time of lighter server traffic.

---

**RLW00004** The query has been idle for [0] seconds.

**Explanation:** The query has been idle for the indicated number of seconds and the **Idle Query Timeouts** warning limit has been reached.

**User response:** Click OK to continue running the query. Click **Abort** to abort the query. Click **Ignore** to ignore the warning.

---

**RLW00005** The query was cancelled because the Idle Query Timeout limit was exceeded.

**Explanation:** The query was cancelled because the Idle Query Timeout limit was exceeded.

**User response:** Attempt to rerun the query at a time when there is less server traffic.

---

**RLW00006** The operation cannot be performed because the processing of LOB data is disabled.

**Explanation:** The operation can not be performed because the LOB Retrieval Option resource limits is disabled.

**User response:** Check your LOB Retrieval Option resource limit by selecting **Resource Limits** from the **View** menu or contact your administrator for more information.

---

**RPT00001** Unable to process a Level component.

**Explanation:** An internal error occurred.

**User response:** Collect support data by selecting Help > Collect Support Data and forward it to customer support.

---

**RPT00002** Report does not contain a main page.

**Explanation:** The visual report does not contain a main page.

**User response:** Make sure that the report contains a main page.

---

**RPT00003** A group expression ":[0]" is not correct.

**Explanation:** The indicated expression refers to a nonexistent column.

**User response:** Ensure that the expression refers to a valid column.
<table>
<thead>
<tr>
<th>RPT00014</th>
<th>BLOB locator was not initialized.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An error occurred while retrieving a BLOB locator.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Reconnect to the repository and try again.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT00015</th>
<th>Query results were not initialized.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The query results were not initialized.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Reconnect to the repository and try again.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT00033</th>
<th>Internal swing renderer error: too early completion.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The line object is incorrect.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Delete the line object and recreate it in the designer.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT00034</th>
<th>Internal swing renderer error: incorrect segment type.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The line object is incorrect.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Delete the line object and recreate it in the designer.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT00035</th>
<th>Internal error in reporter engine.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error occurred.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Collect support data by selecting Help &gt; Collect Support Data and forward it to customer support.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT00037</th>
<th>Invalid arguments was used: row and column should be positive or zero.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>A row or column in the grid is of a length that is less than zero when viewed in the HTML or SWF rendering mode.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Reload the visual project.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT00038</th>
<th>Parameter {0} already exists.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The specified parameter already exists.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Specify a new name for the new parameter.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT00039</th>
<th>Unable to process one or more expressions in the visual dashboard to render its runtime content.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The renderer attempted to render a scene that is undefined.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Reload the visual project.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT00040</th>
<th>A renderer for the selected report format has generated an error. No output contents has been produced.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>A renderer for the selected report format has generated an error. No output contents have been produced.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Try to view the visual project in another rendering mode, or reload the visual project.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT00041</th>
<th>Internal error: unable to create reporter engine.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error occurred.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Collect support data by selecting Help &gt; Collect Support Data and forward it to customer support.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT00042</th>
<th>Unable to parse previously stored visual project data.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The visual project's data is corrupted.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Collect support data by selecting Help &gt; Collect Support Data and forward it to customer support.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT00043</th>
<th>Unable to locate visual dashboard format file.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>{CONFIG_DIR}/converter/repository/world.xml can not be found or read.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Reinstall DataQuant.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT00044</th>
<th>Unable to read visual dashboard format file.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>{CONFIG_DIR}/converter/repository/world.xml can not be found or read.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Reinstall DataQuant.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT00045</th>
<th>Unable to read visual dashboard scene.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The visual project's data is corrupted.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Collect support data by selecting Help &gt; Collect Support Data and forward it to customer support.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT00048</th>
<th>Unable to serialize object {0}.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error occurred.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Collect support data by selecting Help &gt; Collect Support Data and forward it to customer support.</td>
</tr>
</tbody>
</table>
RPT00049  Internal serialization error: no tag for [0] object class.
Explanation: The tag description for the indicated class could not be found during the serialization process.
User response: Reinstall DataQuant.

RPT00050  Unknown parameter type.
Explanation: An unknown parameter type was encountered during the serialization process.
User response: Reload the visual project.

RPT00051  Invalid action type.
Explanation: An unknown action type was encountered while loading the visual project.
User response: Reload the visual project.

RPT00052  Embedded visual dashboard picture is corrupted.
Explanation: An embedded image in the visual dashboard is corrupted and can not be opened.
User response: Check all of the embedded images in the visual dashboard and delete any that are corrupted.

RPT00053  Invalid color value.
Explanation: Corrupted tags were encountered while loading a ColorMap or a ColorSequence.
User response: Reload the visual project.

RPT00054  Invalid threshold value for color map.
Explanation: Corrupted tags were encountered while loading a ColorMap or a ColorSequence.
User response: Reload the visual project.

RPT00055  Invalid event type.
Explanation: An unknown event type was encountered.
User response: Reload the visual project.

RPT00056  Missing image alias.
Explanation: An unknown image was found.
User response: Reload the visual project.

RPT00057  Invalid number value.
Explanation: While loading the visual project, a string that was supposed to contain a number value was encountered without digit characters.
User response: Reload the visual project or try using a different locale.

RPT00058  Invalid date value.
Explanation: While loading the visual project, a string that was supposed to contain a date value was encountered with characters that could not be parsed into a date value.
User response: Reload the visual project or try using a different locale.

RPT00059  Invalid point value.
Explanation: While loading the visual project, a string that was supposed to contain a point value was encountered with characters that could not be parsed into a point value.
User response: Reload the visual project.

RPT00060  Unable to read visual dashboard.
Explanation: An error occurred while loading the visual dashboard.
User response: Reload the visual dashboard.

RPT00061  (Multiple possible messages; see Explanation)
Explanation: 1. Missing query alias. - A query with no query name tag was encountered while loading the visual project.
2. The offline visual report cannot be saved in file. - Offline visual reports can not be saved to a file.
User response: 1. Reload the visual project.
2. Convert the visual report to an online report before saving to a file.

RPT10001  Source Visionary World file has not been specified.
Explanation: The Visionary World file can not be found.
User response: Ensure that the Visionary World file exists and that it can be read.
RPT10002  Configuration directory has not been specified.
Explanations: The configuration directory for the converter can not be found.
User response: Reinstall DataQuant.

RPT10003  Source Visionary World file has not been found.
Explanations: The Visionary World file can not be found.
User response: Ensure that the Visionary World file exists.

RPT10004  Configuration directory has not been found.
Explanations: The configuration directory for the converter can not be found.
User response: Reinstall DataQuant.

RPT10005  Configuration file {} has not been found.
Explanations: The configuration file for the converter can not be found.
User response: Reinstall DataQuant.

RPT10006  Unable to locate Visionary Connection file(s) in the folder {}.
Explanations: The Visionary Connection files do not exist in the proper locations.
User response: Check that all Visionary World files are saved in the correct locations.

RPT10007  Unable to read Visionary Query file {}.
Explanations: The Visionary World file that is specified in the query can not be found.
User response: Check that all Visionary World files are saved in the correct locations.

RPT10008  Unable to locate Visionary Scene file {}.
Explanations: The indicated scene file is not saved in the correct location.
User response: Check that all Visionary World files are saved in the correct locations.

RPT10009  Unable to read an image file {}.
Explanations: The image file that is specified by the indicated name can not be found.
User response: Check that all Visionary World files are saved in the correct locations.

RPT10010  Picture file {} is too large.
Explanations: The indicated picture file is too long.
User response: Specify a smaller picture file to be used in the visual project.

RPT10011  XML parsing error.
Explanations: An XML parsing error occurred while attempting to convert the Visionary World.
User response: Save the Visionary World file in Visionary and try again.

RPT10012  XML reading error.
Explanations: An input/output error occurred while attempting to read the XML data.
User response: Try to convert the XML data again.

RPT10013  Unable to create Visual Dashboard object.
Explanations: An error occurred while attempting to convert the Visionary World file.
User response: Try to convert the Visionary World file again.

RPT10014  Cannot parse configuration file {}.
Explanations: An error occurred while attempting to parse the converter configuration files.
User response: Reinstall DataQuant.

RPT10015  Visual Dashboard for a connection is not specified.
Explanations: The Visionary World file was not loaded properly.
User response: Reload the Visionary World file.

RPT10016  Invalid color description has been found.
Explanations: Corrupted tags were encountered while loading a ColorMap or ColorSequence.
User response: Reload the Visionary World file.

RPT10017  Invalid color map description has been found.
Explanations: Corrupted tags were encountered while loading a ColorMap.
User response: Reload the Visionary World file.
RPT10018  Unknown tag: [0]
Explanation: Unknown tags were encountered while attempting to convert the Visionary World file.
User response: Try to convert the Visionary World file again.

RPT10019  Invalid scene template description.
Explanation: An error occurred while loading the Visionary World scene file.
User response: Reload the Visionary World scene file.

RPT10020  Corrupted property value has been found.
Explanation: An incorrect format or a corrupted value was specified in a property tag.
User response: Reload the Visionary World file.

RPT10021  Invalid URL or path format: "[0]"
Explanation: The indicated URL or path format is invalid.
User response: Check the URL for errors or specify a different URL.

RPT10022  Unknown or unsupported event type
Explanation: A corrupted or unknown event type was encountered.
User response: Reload the Visionary World file.

RPT10023  Function name in expression is missing
Explanation: A function name in an expression is missing.
User response: Reload the Visionary World file.

RPT10024  Cannot parse function argument
Explanation: The function argument could not be parsed while loading the Visionary World file.
User response: Reload the Visionary World file.

RPT10025  Invalid scene template description.
Explanation: An error occurred while loading the Visionary World scene file.
User response: Reload the Visionary World file.

RPT10026  Unable to parse a number: [0]
Explanation: While loading the Visionary World file, a string that was supposed to contain a number value was encountered without digit characters.
User response: Reload the Visionary World file.

RPT10027  PointSet component XML is corrupted.
Explanation: While loading the Visionary World file, a string that was supposed to contain a point value was encountered with characters that could not be parsed into PointSet value.
User response: Reload the Visionary World file.

RPT10028  Unknown expression type [0].
Explanation: An unknown expression type was encountered while loading the Visionary World file.
User response: Reload the Visionary World file.

RPT10029  One or more queries do not have alias.
Explanation: A query alias tag that does not contain a name property was encountered while converting the Visionary World file.
User response: Reload the Visionary World file.

RPT10030  Unable to parse a point value.
Explanation: While loading the Visionary World file, a string that was supposed to contain a point value was encountered with characters that could not be parsed into PointSet value.
User response: Reload the Visionary World file.

RPT10031  Visionary World XML has been corrupted.
Explanation: An error occurred while attempting to load the Visionary World file.
User response: Reload the Visionary World file.

RPT10032  Internal error: unable to access Visionary World.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RPT10033  Unable to add parameter [0] to a Visual Dashboard
Explanation: An internal error occurred.
User response: Collect support data by selecting Help
> Collect Support Data and forward it to customer support.

**RPT20001**  
**Unable to access repository during Visual Report conversion.**  
**Explanation:** The repository could not be accessed while the Visual Report was being converted.  
**User response:** This could possibly be because you were not connected to the repository. Make sure you are connected and try again.

**RPT20002**  
**Source visual report file has not been specified.**  
**Explanation:** The source visual report file has not been specified.  
**User response:** Specify the source visual report file.

**RPT20003**  
**Source visual report file has not been found.**  
**Explanation:** The source visual report file could not be found.  
**User response:** Specify a valid source visual report file.

**RPT20004**  
**Visual Report name has not been specified.**  
**Explanation:** The visual report name has not been specified.  
**User response:** Specify a name for the visual report.

**RPT20005**  
**Unable to create visual report.**  
**Explanation:** An error occurred while creating the visual report.  
**User response:** Reload the visual report.

**RPT20006**  
**Unable to open an embedded image.**  
**Explanation:** An embedded image file was not found.  
**User response:** Reload the visual report.

**RPT20007**  
**Unknown section type: {0}**  
**Explanation:** The structure of a section set and type was not found while loading the visual report.  
**User response:** Reload the visual report.

**RPT20008**  
**Unable to create Visual Report page component.**  
**Explanation:** An error occurred while parsing and creating a page component.  
**User response:** Reload the visual report.

**RPT20009**  
**Internal error: unable to access Visual Report.**  
**Explanation:** An internal error occurred.  
**User response:** Collect support data by selecting Help > Collect Support Data and forward it to customer support.

**RPT20011**  
**Unable to set a line style for a line control.**  
**Explanation:** An undefined line style was encountered while loading the visual report.  
**User response:** Reload the visual report.

**RPT20012**  
**Invalid linked picture alias has been found in Visual Report.**  
**Explanation:** A link to a picture without a name was encountered while parsing the visual report data.  
**User response:** Reload the visual report.

**RPT20013**  
**Unable to create Visual Report control.**  
**Explanation:** An error occurred while loading a primitive or control object.  
**User response:** Reload the visual report.

**RPT20015**  
**Invalid parameter name "(0)" has been found.**  
**Explanation:** The parameter value is invalid.  
**User response:** Specify a different name for the parameter indicated.

**RPT20016**  
**Invalid parameter value "(0)" has been found.**  
**Explanation:** The parameter value is invalid.  
**User response:** Specify a different value for the parameter indicated.

**RPT30001**  
**Navigate to Embedded Scene: Embedded Scene not found for name "(0)".**  
**Explanation:** The Navigate to Embedded Scene action failed because the indicated embedded scene could not be found.  
**User response:** Correct the name of the embedded scene that is associated with the Navigate to Embedded Scene action, or specify another.
<table>
<thead>
<tr>
<th>RPT30002</th>
<th>Navigate back: can't execute exit action (maybe history is empty).</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The Navigate Back action failed, possibly because the navigation history is empty.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Reload the visual project and try again.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT30021</th>
<th>DesignReporter: LevelTemplate is null.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error occurred.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Collect support data by selecting Help &gt; Collect Support Data and forward it to customer support.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT30022</th>
<th>DesignReporter: unknown component {0}.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An unknown component was encountered during runtime rendering.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Reload the visual project.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT30024</th>
<th>The template is invalid. FillStyle or LineStyle property is null!</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An error occurred while running the substitution rule to change the Wedge template.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Check the FillStyle and LineStyle properties for errors.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT30025</th>
<th>The template is invalid. Height == {0}!</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An error occurred while running the substitution rule to change a template.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Check the Height property for errors.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT30026</th>
<th>The template is invalid. Width == {0}!</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An error occurred while running the substitution rule to change a template.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Check the Width property for errors.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT30028</th>
<th>Cannot load resources for package {0} and locale {1}.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The bundle for the resource can not be found.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Restart the application. If this does not fix the problem, reinstall the application.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT30029</th>
<th>Request for resource string &quot;{0}&quot; from package {1} failed.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An error occurred while requesting a string resource.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Restart the application. If this does not fix the problem, reinstall the application.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT30035</th>
<th>No OutputComponentCreator in map for {0}.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An internal error occurred.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Collect support data by selecting Help &gt; Collect Support Data and forward it to customer support.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT30038</th>
<th>Could not find {0}.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>One of the objects in the Connections folder could not be found.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Reload the project or delete the connection and try again.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT30040</th>
<th>Chart cannot read query result set.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An error occurred while creating a ClusterGraph, OrganizationChart, or TreeChart.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Reconnect to the repository and reload the visual project.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT30042</th>
<th>Error - axisLength is infinite. totalPercentages:[0] m_sceneRuntimePanel.getZoomFactor() [1]</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The value of an axis's PercentageLength property is invalid.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Check the PercentageLength property value of the axis for errors.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT30044</th>
<th>Unknown checkbox value: {0}.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>A value other than UNCHECKED, CHECKED, or INDETERMINATE was specified.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Ensure that any action that sets the value of a checkbox sets it to UNCHECKED, CHECKED, or INDETERMINATE.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT30045</th>
<th>Unknown m_style: {0}.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An invalid style property was specified for a ComboBox object.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Check the style property for the ComboBox object for errors.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RPT30047</th>
<th>Line is not horizontal or vertical.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>A segment in an ElbowConnector object is set to neither vertical or horizontal.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Reload the visual project.</td>
</tr>
</tbody>
</table>
Unsupported segment type: {0}.

Explanation: An unsupported segment type was encountered for an ElbowConnector object.

User response: Reload the visual project.

Could not get corner type last: {0} start: {1} end: {2}.

Explanation: The start and end coordinates of a line segment do not define a corner in an ElbowConnector object.

User response: Reload the visual project, or delete and re-create the ElbowConnector object.

Should not have gotten here.

Explanation: An internal error occurred.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

Unknown horizontal align: {0}.

Explanation: A value other than LEFT, RIGHT, or CENTER was encountered in the HorizontalAlign property of an object.

User response: Ensure that any action that sets the value of the HorizontalAlign property of an object sets it as LEFT, RIGHT, or CENTER.

Unknown vertical align: {0}.

Explanation: A value other than TOP, BOTTOM, or CENTER was encountered in the VerticalAlign property of an object.

User response: Ensure that any action that sets the value of the VerticalAlign property of an object sets it as TOP, BOTTOM, or CENTER.

Unknown location anchor: {0}.

Explanation: A value other than BOTTOMLEFT, LEFTCENTER, TOPLEFT, BASELINELEFT, CENTER, TOPCENTER, BASELINECENTER, BOTTOMRIGHT, RIGHTCENTER, or BASELINERIGHT was encountered in the LocationAnchor property of an object.

User response: Ensure that any action that sets the value of the LocationAnchor property of an object sets it as BOTTOMLEFT, LEFTCENTER, TOPLEFT, BASELINELEFT, BOTTOMCENTER, CENTER, TOPCENTER, BASELINECENTER, BOTTOMRIGHT, RIGHTCENTER, or BASELINERIGHT.

Unknown arrowheadStyle: {0}.

Explanation: A value other than NONE, TRIANGLE, CIRCLE, DIAMOND, or SWEPTPOINTER was encountered in the MarkerShape property of an arrow object.

User response: Ensure that any action that sets the value of the MarkerShape property of an arrow object sets it as NONE, TRIANGLE, CIRCLE, DIAMOND, or SWEPTPOINTER.

Unknown marker shape: {0}.

Explanation: A value other than HOLLOWCIRCLE, HOLLOWDIAMOND, HOLLOW SQUARE, HOLLOW TRIANGLE, HOLLOW TRIANGLE2, HOLLOWSTAR, HOLLOWPLUS, SOLIDCIRCLE, SOLIDDIAMOND, SOLIDSQUARE, SOLIDTRIANGLE, SOLID TRIANGLE2, SOLIDSTAR, SOLIDPLUS, CROSS, PLUS, BOXCROSS, or BOXPLUS was encountered in the MarkerShape property of a data point object.

User response: Ensure that any action that sets the value of the MarkerShape property of a data point object sets it as HOLLOWCIRCLE, HOLLOWDIAMOND, HOLLOW SQUARE, HOLLOW TRIANGLE, HOLLOW TRIANGLE2, HOLLOWSTAR, HOLLOWPLUS, SOLIDCIRCLE, SOLIDDIAMOND, SOLIDSQUARE, SOLIDTRIANGLE, SOLID TRIANGLE2, SOLIDSTAR, SOLIDPLUS, CROSS, PLUS, BOXCROSS, or BOXPLUS.

Unknown attachmentStyle: {0}.

Explanation: A value other than TOPLEFT, TOPCENTER, TOP, TOPRIGHT, CENTERLEFT, CENTER, CENTERRIGHT, BOTTOMLEFT, BOTTOMCENTER, BOTTOM, or BOTTOMRIGHT was encountered in the AttachmentStyle property of a connector object.

User response: Ensure that any action that sets the value of the AttachmentStyle property of a connector object sets it as TOPLEFT, TOPCENTER, TOP, TOPRIGHT, CENTERLEFT, CENTER, CENTERRIGHT, BOTTOMLEFT, BOTTOMCENTER, BOTTOM, or BOTTOMRIGHT.

Unknown border type for table: {0}.

Explanation: An invalid value was encountered in the BorderType property of an object.

User response: Ensure that any action that sets the BorderType property of an object sets a valid value.

Unknown m_horizontalAlign: {0}.

Explanation: An invalid value was encountered in the HorizontalAlign property of a table cell.

User response: Ensure that any action that sets the HorizontalAlign property of a table cell sets a valid value.
RPT30063  Unknown m_verticalAlign: [0].
Explanation: An invalid value was encountered in the VerticalAlign property of a table cell.
User response: Ensure that any action that sets the VerticalAlign property of a table cell sets a valid value.

RPT30064  Unknown border type for table: [0].
Explanation: An invalid value was encountered in the Border property of a table.
User response: Ensure that any action that sets the Border property of a table sets a valid value.

RPT30065  Unknown scrollBarStyle: [0].
Explanation: An invalid value was encountered in the ScrollBarStyle property of an object.
User response: Ensure that any action that sets the ScrollBarStyle property of an object sets a valid value.

RPT30066  Unknown AlignmentStyle: [0].
Explanation: An invalid value was encountered in the AlignmentStyle property of a textbox object.
User response: Ensure that any action that sets the AlignmentStyle property of a textbox object sets a valid value.

RPT30068  Unknown lineStylePattern: [0].
Explanation: An invalid value was encountered in the LineStylePattern property of an object.
User response: Ensure that any action that sets the LineStylePattern property of an object sets a valid value.

RPT30069  Unknown gradient type: [0].
Explanation: An invalid value was encountered in the GradientType property of an object.
User response: Ensure that any action that sets the GradientType property of an object sets a valid value.

RPT30070  Unknown borderType: [0].
Explanation: An invalid value was encountered in the BorderType property of a textbox or embedded scene object.
User response: Ensure that any action that sets the BorderType property of a textbox or embedded scene object sets a valid value.

RPT30071  Unknown imageAnchor: [0].
Explanation: An invalid value was encountered in the ImageAnchor property of an object.
User response: Ensure that any action that sets the ImageAnchor property of an object sets a valid value.

RPT30072  Embedded Scene: Unknown scrollBarStyle.
Explanation: An invalid value was encountered in the ScrollBarStyle property of an embedded scene object.
User response: Ensure that any action that sets the ScrollBarStyle property of an embedded scene object sets a valid value.

RPT30073  Embedded Scene: unknown typeOfFind: [0].
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RPT30076  The cell chosen by client not found.
Explanation: The action can not be performed when using the HTML rendering mode.
User response: Reload the visual project.

RPT30077  Grid cell's click event could not be dispatched.
Explanation: The action can not be performed when using the HTML rendering mode.
User response: Reload the visual project.

RPT30079  Unknown page type - [0].
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer service.

RPT30082  Parameters group cloning problem [0].
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RPT30083  No class in map for [0].
Explanation: An internal error occurred.
User response: Collect support data by selecting Help
Collect Support Data and forward it to customer support.

RPT30088  Unable to serialize an object: {0}.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help
> Collect Support Data and forward it to customer support.

RPT30089  Value must be positive: {0}.
Explanation: The indicated value must be a positive number.
User response: Ensure that the indicated value is a positive number.

RPT30090  Level was null when zoom was {0}.
Explanation: No level is specified for the indicated zoom percentage.
User response: Ensure that a level is specified for the indicated zoom percentage.

RPT30091  Chart can't find parent column: {0}.
Explanation: The indicated parent column could not be found in the result set.
User response: Ensure that a valid column named is specified in the DataLocator property of the group.

RPT30092  Chart can't find child column: {0}.
Explanation: The indicated child column could not be found in the result set.
User response: Ensure that a valid column named is specified in the DataLocator property of the group.

RPT30095  Hierarchy chart must have one DataTemplate child.
Explanation: A hierarchy chart must have one DataTemplate child.
User response: Check the Project Explorer and ensure that the hierarchy chart only has one DataTemplate child.

RPT30096  CurrentRowIndex < 0.
Explanation: The specified row index is less than zero. The row index must be an integer that is equal to or greater than zero.
User response: Ensure the specified row index is an integer that is equal to or greater than zero.

RPT30097  Unknown interval: {0}.
Explanation: An invalid value was specified for the DateAxis property of an object.
User response: Ensure that any action that sets the value of the DateAxis property of an object sets a valid value.

RPT30098  Unhandled interval: {0}.
Explanation: An invalid value was specified for the DateAxis property of an object.
User response: Ensure that any action that sets the value of the DateAxis property of an object sets a valid value.

RPT30099  Unknown dateInterval: {0}.
Explanation: An invalid value was specified for the DateAxis property of an object.
User response: Ensure that any action that sets the value of the DateAxis property of an object sets a valid value.

RPT30104  Shouldn't get here - roundIncrement: {0} power: {1}.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help
> Collect Support Data and forward it to customer support.

RPT30105  Group name is undefined for group header.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help
> Collect Support Data and forward it to customer support.

RPT30106  Group name is undefined for group footer.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help
> Collect Support Data and forward it to customer support.

RPT30107  m_templateRows contained a {0}.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help
> Collect Support Data and forward it to customer support.
RPT30109  Embedded Scene: Linked not found for name {0}.
Explanation: The value of the LinkedScene property of an embedded scene is invalid.
User response: Ensure that the scene that the embedded scene is linked to exists and that the link text is correct.

RPT30111  Unable to clone component {0}.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RPT30112  Property {0} of component {1} doesn't appear to be of type {2}.
Explanation: The property of the indicated object is of an invalid type.
User response: Ensure that any action or expression that sets the value of the property sets it to a valid type.

RPT30114  Swing output cannot be created for component {0}.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RPT30117  Component {0} does not support action {1}.
Explanation: The indicated component does not support the indicated action.
User response: Specify a different action for the component and try again.

RPT30118  Component {0} does not support property {1}.
Explanation: The indicated component does not support the indicated property.
User response: Specify a different property and try again.

RPT30120  (Multiple possible messages; see Explanation)
Explanation:
1. The global parameter "[0]" does not exist - The indicated global parameter does not exist.
2. Query "[0]" does not contain column named "[1]" - The indicated column does not exist in the indicated query.

User response:
1. Check the global parameter name for errors or specify a different global parameter.
2. Check the column name for errors or specify a different column name.

RPT30121  Result set index "[0]" for query "[1]" is out of bounds.
Explanation: An invalid value was set for the ResultSetIndex of a layout object.
User response: Ensure that any action that sets the ResultSetIndex property of a layout object sets a valid value.

RPT30122  Query with key "[0]" does not exist.
Explanation: A query with the indicated key does not exist in the repository.
User response: Select the query in the Repository Explorer or Workspaces view and copy and paste the key value from the Properties view, or specify a different query's key.

RPT30123  Component ")[0]\" cannot be created due to the following problem: {1}.
Explanation: The indicated component can not be created due to the indicated error.
User response: Resolve the indicated error and try again.

RPT30124  Result set index cannot be defined.
Explanation: An internal error occurred.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RPT30126  Security list "[0]" cannot be found.
Explanation: The indicated security list can not be found.
User response: Check the security list name for errors or specify a different security list.

RPT30127  Visual project ")[0]\" cannot be loaded because of missing project descriptor.
Explanation: The indicated visual project can not be loaded because it is missing a project descriptor.
User response: Ensure that the visual project has a project descriptor and retry to load the project.
RSE01000 [0]
Explanation: The indicated error occurred while accessing the repository security provider.
User response: Resolve the indicated error and try again.

RSE04000 System internal error occurred, please check the error log.
Explanation: A system internal error occurred.
User response: Check the error log by selecting Output from the context menu of the Window menu.

RSE04001 Cannot find User [0]
Explanation: The indicated user can not be found.
User response: Check the name of the user for errors and try again.

RSE04002 User [0] already exists;
Explanation: The indicated user already exists.
User response: Specify a different user.

RSE04003 Group [0] already exists;
Explanation: The indicated group already exists.
User response: Specify a different group.

RSE04004 Cannot find group [0]
Explanation: The indicated group can not be found.
User response: Specify a different group.

RSE04005 Not logged in with security provider.
Explanation: You attempted to access the repository without logging in.
User response: Login to the repository and try again.

RSE04006 The user ID and password are invalid
Explanation: The user ID and password that were entered are invalid.
User response: Check the user ID and password for errors and try again.

RSE04007 Name "[0]" is in conflict with the existing name.
Explanation: The new name that was entered already exists.
User response: Select a different name.

RSE04008 Object access denied
Explanation: Access to the specified object is denied.
User response: Select a different object to access or contact your administrator for more information.

RSE04011 Cannot remove User [0] from Group [1]
Explanation: The indicated user can not be removed from the indicated group.
User response: Select another user to be removed from the group.

RSE04012 Cannot delete User [0]
Explanation: The indicated user can not be deleted.
User response: Select another user to be deleted.

RSE04013 Cannot delete Group [0]
Explanation: The indicated group can not be deleted.
User response: Select another group to be deleted.

RSE04014 Cannot initialize security provider,[0]
Explanation: The indicated error occurred while initializing the repository.
User response: Resolve the indicated error and try again.

RSE04015 This security provider is a read-only one.
Explanation: You can not change the principals of a read-only security provider.
User response: Choose another security provider to modify.

RSE04016 Cannot find any name associates with UID [0] in Repository
Explanation: The indicated repository identifier is invalid.
User response: Specify a valid repository identifier and try again.

RSE04017 AuthType is not anonymous("none"),please specify principal and credentials
Explanation: An error occurred while initializing the LDAP security provider.
User response: Change the repository initialization parameters and try again.
RSE04018 LDAP Configuration contains illegal values or some required values are missing, please check the preferences.

Explanation: An error occurred while initializing the LDAP security provider.
User response: Change the repository initialization parameters and try again.

RSE04019 Cannot connect to the LDAP server. Communication broken or Directory Service is not available.

Explanation: An error occurred while connecting to the LDAP directory.
User response: Check the network connection and try again.

RSE04020 Cannot read User's password attribute (Make sure you have enough permission or PasswordAttribute in configuration is correct.)

Explanation: An error occurred while reading data from the LDAP directory.
User response: Ensure that the LDAP service is accessible.

RSE04021 Invalid search filter in LDAP configuration

Explanation: The specified search filter in the LDAP configuration is invalid.
User response: Specify a valid search filter for the LDAP configuration.

RSE04022 LDAP server returned error

Explanation: An error occurred while reading data from the LDAP directory.
User response: Ensure that the LDAP service is accessible.

RSE04023 Cannot find the specified name [0]

Explanation: The indicated user name can not be found.
User response: Specify a different user name.

RSE04024 Not enough permission to perform the LDAP operations.

Explanation: You user ID does not have permission to perform LDAP operations.
User response: Contact your administrator for more information concerning specific permissions and allowances.

RSE04025 Cannot find a attribute of entry [0] specified in LDAP configurations.

Explanation: An error occurred while reading attribute data from the LDAP configuration.
User response: Correct the configuration parameters and try again.

RSE04026 UUID [0] mapping to an invalid distinguished name [1].(2)

Explanation: The indicated distinguished name is invalid.
User response: Specify a valid distinguished name and try again.

RSE04028 Incorrect operation.

Explanation: An error occurred while editing the principals of the security provider.
User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

RSE04029 Cannot register security provider.

Explanation: An error occurred while registering the security provider in the provider registry.
User response: Ensure that the registered name of the provider is unique.

RSE04030 Security provider not initialized.

Explanation: An error occurred while accessing the security provider service because the service is not initialized.
User response: Initialize the security provider and try again.

RSE04031 Security provider not registered.

Explanation: An error occurred while accessing the security provider service because the security provider is not registered.
User response: Register the security provider and try again.

RSE04032 You can not remove default Administrator.

Explanation: The default administrator can not be deleted.
User response: Specify a different administrator to be deleted.
<table>
<thead>
<tr>
<th>Code</th>
<th>Message</th>
<th>Explanation</th>
<th>User response</th>
</tr>
</thead>
<tbody>
<tr>
<td>RSE04033</td>
<td>You cannot change the default Administrator's membership.</td>
<td>The membership of the default administrator cannot be changed.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Explanation:</strong> The membership of the default administrator cannot be changed.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>User response:</strong> Specify the membership of a different administrator to change.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RSE04035</td>
<td>You cannot rename the Administrators group.</td>
<td>The group of the default administrator cannot be renamed.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Explanation:</strong> The group of the default administrator cannot be renamed.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>User response:</strong> Specify the group of a different administrator to be renamed.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RSE04036</td>
<td>Password has expired and is no longer valid.</td>
<td>The specified password has expired and is no longer valid.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Explanation:</strong> The specified password has expired and is no longer valid.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>User response:</strong> Specify an updated password and try again.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RSE04037</td>
<td>Group could not became its own parent.</td>
<td>A group may not be its own parent. You must select a different group as the parent.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Explanation:</strong> A group may not be its own parent. You must select a different group as the parent.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>User response:</strong> Select a different group to be the parent of the specified group.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SPL01001</td>
<td>Cannot find objectClass attribute of {0}</td>
<td>The indicated object does not have an objectClass attribute. Every object in the LDAP directory must contain this attribute.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Explanation:</strong> The indicated object does not have an objectClass attribute. Every object in the LDAP directory must contain this attribute.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>User response:</strong> Specify an objectClass attribute for the indicated object.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SPL01002</td>
<td>The returned entry {0} is not acceptable for LDAP Security Provider, its objectClass is not (1) or (2)</td>
<td>The indicated object is defined as neither a user or group in the LDAP security provider.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Explanation:</strong> The indicated object is defined as neither a user or group in the LDAP security provider.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>User response:</strong> Edit the objectClass attribute of the indicated object to specify whether it is a user or group.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SPL01003</td>
<td>Cannot read the Group's name (attribute: {0})</td>
<td>The LDAP directory does not contain an object that has the required attribute.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Explanation:</strong> The LDAP directory does not contain an object that has the required attribute.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>User response:</strong> Edit the groupName attribute so that it will point to an existing object in the LDAP directory.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SPL01004</td>
<td>Cannot read the User's name (attribute: {0})</td>
<td>The LDAP directory does not contain an object that has the required attribute.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Explanation:</strong> The LDAP directory does not contain an object that has the required attribute.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>User response:</strong> Edit the userName attribute so that it will point to an existing object in the LDAP directory.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SPL01005</td>
<td>LDAP server host or port was not specified.</td>
<td>The LDAP server host or port was not specified.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Explanation:</strong> The LDAP server host or port was not specified.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>User response:</strong> Specify the LDAP server and host.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SPL01006</td>
<td>The base distinguish name of the LDAP server must be present.</td>
<td>The base distinguished name of the LDAP server was not specified.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Explanation:</strong> The base distinguished name of the LDAP server was not specified.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>User response:</strong> Specify the base distinguished name of the LDAP server.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SPL01007</td>
<td>The distinguish name of user principal must be present.</td>
<td>The distinguished name of the principal user was not specified.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Explanation:</strong> The distinguished name of the principal user was not specified.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>User response:</strong> Specify the distinguished name of the principal user or use anonymous authentication.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SPL01008</td>
<td>The credentials of user entry must be specified.</td>
<td>The credentials of the entry user were not specified.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Explanation:</strong> The credentials of the entry user were not specified.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>User response:</strong> Specify the credentials of the entry user or use anonymous authentication.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SPL01009</td>
<td>Cannot create LDAP entry for administrator user: {0}</td>
<td>The user who is associated with the administrative account could not be found in the LDAP directory.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Explanation:</strong> The user who is associated with the administrative account could not be found in the LDAP directory.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>User response:</strong> Specify the administrative user of the LDAP directory.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SPL01010</td>
<td>Cannot create LDAP entry for administrators group: {0}</td>
<td>The group that is associated with the administrative account could not be found in the LDAP directory.</td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>Explanation:</strong> The group that is associated with the administrative account could not be found in the LDAP directory.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><strong>User response:</strong> Specify the administrative group of the LDAP directory.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SPL0101</td>
<td>Cannot create LDAP entry for everyone group: {0}</td>
<td></td>
<td></td>
</tr>
<tr>
<td>------------</td>
<td>--------------------------------------------------</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Explanation:</strong></td>
<td>The group that is associated with the everyone account could not be found in the LDAP directory.</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Specify the everyone group of the LDAP directory.</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SPL0102</th>
<th>Cannot create LDAP entry for root group: {0}</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The object that is associated with the Base DN could not be found in the LDAP directory.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Specify the Base DN object in the LDAP directory.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SPL0103</th>
<th>Cannot find the JNDI extension package or Sun's LDAP Service Provider.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The JNDI extension package or Sun's LDAP Service Provider was not located in the CLASSPATH.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Ensure that the CLASSPATH includes both the JNDI extension package and Sun's LDAP Service Provider.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SPL0104</th>
<th>The catalog object is missing.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The catalog object is not set.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Specify the catalog object and try again.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SPL0105</th>
<th>Empty name is not allowed.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>No name was specified as an RDN.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Specify a name as an RDN.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SPL0106</th>
<th>Illegal RDN of security object (offending string: &quot;{0}&quot;)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The RDN is invalid. The RDN must be one or more LDAP components that are separated by commas.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Specify valid components for the RDN.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SPL0107</th>
<th>Unknown attribute name &quot;{0}&quot; in RDN.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>An unknown attribute name was encountered while parsing the RDN.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Check the attributes of the RDN for errors.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SPL0108</th>
<th>Currently Client Authentication is not supported.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>Client Authentication is not supported.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Use anonymous authentication.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SPL0109</th>
<th>SSL is not correctly initialized.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The secure socket protocol factory is not initialized.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Restart the application and try again.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SPL0110</th>
<th>Attribute {0} is absent.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The indicated attribute was not found while reading the LDAP object.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Specify a valid value for the indicated attribute.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SPL0111</th>
<th>Object class name &quot;{0}&quot; is unknown.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The object name has an invalid class name; it must be either qmfUser or qmfGroup.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Specify either qmfUser or qmfGroup for the indicated object name.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SQM02001</th>
<th>An input/output error occurred while processing package &quot;{0}&quot;. {1}</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The indicated error occurred while processing the indicated package.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Resolve the indicated error and try again.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SQM02002</th>
<th>Error in SQLJ customization: {0}</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The indicated error occurred while processing the SQLJ customization.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Resolve the indicated error and try again.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SQM02003</th>
<th>No SQLJ installed</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The SQLJ library could not be loaded.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Add sqlj.zip to the driver definition.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SQM02004</th>
<th>An error occurred while retrieving profile data for package &quot;{0}&quot;. {1}</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The indicated error occurred while the profile data for the indicated package was being loaded.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Resolve the indicated error and try again.</td>
</tr>
</tbody>
</table>
SQM02005  An error occurred while loading the profile for package "(0)".

Explanation: The indicated error occurred while the profile for the indicated package was being loaded.

User response: Resolve the indicated error and try again.

SQM02006  While loading the class with the implementation of package "(0)" the following error has occurred: "(1)".

Explanation: The indicated error occurred while loading class with the implementation of the indicated package was being loaded.

User response: Reinstall the application and try again.

SQM02007  A database access error occurred while processing package "(0)". (1)

Explanation: The indicated database access error occurred while the package was being processed.

User response: Resolve the indicated database access error and try again.

SQM02008  A database access error occurred: (1)

Explanation: The indicated database access error occurred.

User response: Resolve the indicated error and try again.

SQM02009  User has no rights to execute queries of package "(0)". Original error follows: (1)

Explanation: The user does not have permission to access the package because of the indicated reason.

User response: Contact your database administrator.

SQM02010  An error occurred while deleting profile for package "(0)". (1)

Explanation: The indicated error occurred while the profile for the indicated package was being deleted.

User response: Resolve the indicated error and try again.

SQM02011  Empty customizer call pattern

Explanation: Package customization failed because of an unexpected error.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

SQM02013  An error occurred while loading data for package "(0)".

Explanation: An error occurred while the data for the indicated package was being loaded.

User response: Resolve the indicated error and try again.

SQM02014  Wrong customizer call pattern: (0)

Explanation: Package customization failed because of an unexpected error.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

SQM02016  Cannot create collection "(0)" due to the following: (1)

Explanation: The indicated collection could not be created due to the indicated error.

User response: Resolve the indicated error and try again.

SQM02017  Profile for package "(0)" is too long to be stored at database

Explanation: The profile for the indicated package is too long to be stored at the database.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

SQM02018  No collection ID (profile owner) is specified. Cannot proceed in static mode.

Explanation: The SQLJ package can not be loaded if no collection ID is specified.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

SQM02019  Cannot prepare statement "(0)" of package "(1)" due to the following error: "(2)"

Explanation: The indicated statement of the package can not be prepared due to the indicated error.

User response: Resolve the indicated error and try again.
While checking user rights on package "{0}" the following error has occurred: "{1}"

Explanation: The indicated error occurred while the user rights on the indicated package were checked.

User response: Resolve the indicated error and try again.

Inconsistent profile data for queries package "{0}" were detected. Rebind package.

Explanation: Inconsistent profile data for the indicated queries package was detected.

User response: Rebind the package.

The required parameter "{0}" is missing.

Explanation: The indicated required parameter is missing.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

An error occurred while initializing the connection. {0}

Explanation: The indicated error occurred while the connection was initialized.

User response: Resolve the indicated error and try again.

{0}

Explanation: The indicated database access error occurred.

User response: Resolve the indicated error and try again.

An error occurred while loading package description class "{0}". {1}

Explanation: The indicated error occurred while the package description class was loaded.

User response: Resolve the indicated error and try again.

An error occurred while loading the profile for package "{0}". {1}

Explanation: The indicated error occurred while the profile for the indicated package was loaded.

User response: Resolve the indicated error and try again.

Static mode profile for package "{0}" is not found.

Explanation: The static mode for the indicated package was not found.

User response: Reinstall the application and try again.

An error occurred while creating directory "{0}".

Explanation: The temporary directory for the customization process could not be created.

User response: Ensure that the application has permissions to create the directory and try again.

An error occurred while creating profiles directory "{0}".

Explanation: An error occurred while the indicated profile's directory was created.

User response: Ensure that the application has permissions to create the directory and try again.

An error occurred while saving the profile for package "{0}". {1}

Explanation: The indicated error occurred while the profile for the indicated package was saved.

User response: Resolve the error and try again.

An error occurred while saving the packages description file. {0}

Explanation: The indicated error occurred while the package description file was saved.

User response: Resolve the error and try again.

An error occurred while starting customization. {0}

Explanation: The indicated error occurred while customization was started.

User response: Resolve the error and try again.

An error occurred while customizing package "{0}". {1}

Explanation: The indicated error occurred while the indicated package was customized.

User response: Resolve the indicated error and try again.
<table>
<thead>
<tr>
<th>SQM02112</th>
<th>An error occurred while creating the profiles data table. {0}</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Explanation:</strong></td>
<td>The indicated error occurred while the data table of the profile was created.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Resolve the indicated error and try again.</td>
</tr>
<tr>
<td>SQM02113</td>
<td>No user ID list for GRANT/REVOKE operation was specified.</td>
</tr>
<tr>
<td><strong>Explanation:</strong></td>
<td>No user ID list for GRANT/REVOKE operation was specified.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Specify a user ID list for the GRANT/REVOKE operation.</td>
</tr>
<tr>
<td>SQM02115</td>
<td>An error occurred while uninstalling the packages. {0}</td>
</tr>
<tr>
<td><strong>Explanation:</strong></td>
<td>The indicated error occurred while the packages were uninstalled.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Resolve the indicated error and try again.</td>
</tr>
<tr>
<td>SQM02116</td>
<td>File with packages description was not found. {0}</td>
</tr>
<tr>
<td><strong>Explanation:</strong></td>
<td>The indicated error occurred while preparing the packages descriptions based on the previously build packages description file.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Collect support data by selecting Help &gt; Collect Support Data and forward it to customer support.</td>
</tr>
<tr>
<td>SQM02117</td>
<td>An error occurred while reading the packages description file. {0}</td>
</tr>
<tr>
<td><strong>Explanation:</strong></td>
<td>The indicated error occurred while the packages description file was being read.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Collect support data by selecting Help &gt; Collect Support Data and forward it to customer support.</td>
</tr>
<tr>
<td>SQM02119</td>
<td>An error occurred while installing the package {0}. {1}</td>
</tr>
<tr>
<td><strong>Explanation:</strong></td>
<td>The indicated error occurred when the indicated package was being installed.</td>
</tr>
<tr>
<td><strong>User response:</strong></td>
<td>Resolve the indicated error and try again.</td>
</tr>
<tr>
<td>SQM02120</td>
<td>An error occurred while uninstalling the package {0}. {1}</td>
</tr>
<tr>
<td><strong>Explanation:</strong></td>
<td>The indicated error occurred when the indicated package was being uninstalled.</td>
</tr>
</tbody>
</table>

**User response:** Resolve the indicated error and try again.

SYNERRCD0001  DSS header length less than 6.
**Explanation:** An internal error in the driver code occurred.
**User response:** Send an error report to the developers.

SYNERRCD0002  DSS header length does not match the number of bytes of data found.
**Explanation:** An internal error in the driver code occurred.
**User response:** Send an error report to the developers.

SYNERRCD0003  DSS header C-byte not D0.
**Explanation:** An internal error in the driver code occurred.
**User response:** Send an error report to the developers.

SYNERRCD0004  DSS header f-bytes either not recognized or not supported.
**Explanation:** An internal error in the driver code occurred.
**User response:** Send an error report to the developers.

SYNERRCD0005  DSS continuation specified but not found.
**Explanation:** An internal error in the driver code occurred.
**User response:** Send an error report to the developers.

SYNERRCD0006  DSS chaining specified but no DSS found.
**Explanation:** An internal error in the driver code occurred.
**User response:** Send an error report to the developers.

SYNERRCD0007  Object length less than four.
**Explanation:** An internal error in the driver code occurred.
**User response:** Send an error report to the developers.

SYNERRCD0008  Object length does not match the number of bytes of data found.
**Explanation:** An internal error in the driver code occurred.
**User response:** Send an error report to the developers.
SYNERRCD0009 • SYNERRCD0024

SYNERRCD0009  Object length greater than maximum allowed.
Explanation: An internal error in the driver code occurred.
User response: Send an error report to the developers.

SYNERRCD0010  Object length less than minimum required.
Explanation: An internal error in the driver code occurred.
User response: Send an error report to the developers.

SYNERRCD0011  Object length not allowed.
Explanation: An internal error in the driver code occurred.
User response: Send an error report to the developers.

SYNERRCD0012  Incorrect large object extended length field.
Explanation: An internal error in the driver code occurred.
User response: Send an error report to the developers.

SYNERRCD0013  Object codepoint index not supported.
Explanation: An internal error in the driver code occurred.
User response: Send an error report to the developers.

SYNERRCD0014  Required object not found.
Explanation: An internal error in the driver code occurred.
User response: Send an error report to the developers.

SYNERRCD0015  Too many command data objects sent.
Explanation: An internal error in the driver code occurred.
User response: Send an error report to the developers.

SYNERRCD0017  Too few command data objects sent.
Explanation: An internal error in the driver code occurred.
User response: Send an error report to the developers.

SYNERRCD0018  Duplicate object present.
Explanation: An internal error in the driver code occurred.
User response: Send an error report to the developers.

SYNERRCD0019  Invalid request correlator specified.
Explanation: An internal error in the driver code occurred.
User response: Send an error report to the developers.

SYNERRCD0020  Required value not found.
Explanation: An internal error in the driver code occurred.
User response: Send an error report to the developers.

SYNERRCD0021  Reserved value not allowed.
Explanation: An internal error in the driver code occurred.
User response: Send an error report to the developers.

SYNERRCD0022  DSS continuation less than or equal to two.
Explanation: An internal error in the driver code occurred.
User response: Send an error report to the developers.

SYNERRCD0023  Objects not in required order.
Explanation: An internal error in the driver code occurred.
User response: Send an error report to the developers.

SYNERRCD0024  DSS chaining bit not 1, but DSSFMT bit 3 set to 1.
Explanation: An internal error in the driver code occurred.
User response: Send an error report to the developers.
<table>
<thead>
<tr>
<th>Error Code</th>
<th>Description</th>
<th>Explanation</th>
<th>User Response</th>
</tr>
</thead>
<tbody>
<tr>
<td>SYNERRCD0025</td>
<td>Previous DSS indicated current DSS has the same request correlator, but the request correlators are not the same.</td>
<td>An internal error in the driver code occurred.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>SYNERRCD0026</td>
<td>Previous DSS indicated current DSS has the same request correlator, but the request correlators are not the same.</td>
<td>An internal error in the driver code occurred.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>SYNERRCD0027</td>
<td>DSS chaining bit not 1, but error continuation requested.</td>
<td>An internal error in the driver code occurred.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>SYNERRCD0028</td>
<td>Mutually-exclusive parameter values specified.</td>
<td>An internal error in the driver code occurred.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>SYNERRCD0029</td>
<td>Codepoint not valid command.</td>
<td>An internal error in the driver code occurred.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>SYNERRCD0030</td>
<td>The atmind instance variable is not set to its default value of 0x00 on an EXCSQLSTT command within an atomic EXCSQLSTT chain.</td>
<td>An internal error in the driver code occurred.</td>
<td>Send an error report to the developers.</td>
</tr>
<tr>
<td>THM1002</td>
<td>Cannot extract main theme CSS file from the archive.</td>
<td>The specified archive does not contain the main CSS file.</td>
<td>Ensure that the main CSS file is saved to the archive and try again.</td>
</tr>
<tr>
<td>THM1003</td>
<td>Already registered. Try another theme name.</td>
<td>A theme with the same name already exists.</td>
<td>Specify a different name for the theme.</td>
</tr>
<tr>
<td>UIM12001</td>
<td>Internal error has occurred: {0}</td>
<td>The indicated internal error occurred.</td>
<td>Resolve the indicated error and try again.</td>
</tr>
<tr>
<td>UIM12004</td>
<td>An input/output error occurred: {0}</td>
<td>The indicated input/output error occurred.</td>
<td>Resolve the indicated error and try again.</td>
</tr>
<tr>
<td>UIM12005</td>
<td>The file does not exist: {0}</td>
<td>The indicated file does not exist.</td>
<td>Specify a valid file and try again.</td>
</tr>
<tr>
<td>VAL00001</td>
<td>Variable {0} not found.</td>
<td>The indicated variable was not found.</td>
<td>Specify a valid variable.</td>
</tr>
<tr>
<td>VAL00003</td>
<td>Type conversion error. Object of {0} type cannot be converted to {1} type.</td>
<td>The indicated object can not be converted to the indicated type.</td>
<td>Specify an object of a different type to be converted, or convert the object to a different type.</td>
</tr>
</tbody>
</table>
VAL00004  Unit conversion error. Unit [0] cannot be converted to [1].

Explanation: The indicated unit can not be converted to the indicated type.

User response: Specify a different unit to be converted, or convert the unit to a different type.

VAL00005  An expression contains a syntax error.

Explanation: An expression that you entered contains a syntax error.

User response: Check the expressions that you typed for syntax errors.

VAL00006  Wrong expression has been used.

Explanation: An incorrect expression has been used.

User response: Specify a valid expression.

VAL00007  Name "[0]" has invalid character at [1]. The name must start with a letter and should not contain special characters. See the help for a list of invalid characters.

Explanation: The indicated name contains an invalid character at the indicated column.

User response: Check the help for syntax rules and retype the invalid character.

VAL00008  Name cannot be empty. The name must start with a letter and should not contain special characters. See the help for a list of invalid characters.

Explanation: The name of a control can not be empty.

User response: Specify a correct name for the control.

VAL00009  Unable to process URL: "[0]"

Explanation: The indicated URL could not be processed.

User response: Check the URL for errors and try again.

VAL00010  Unable to process URI: "[0]"

Explanation: The indicated URI could not be processed.

User response: Check the URI for errors and try again.

VAL00019  Cycle has been found in definition of following properties: [0].

Explanation: The expression contains a cyclic reference.

User response: Edit the expression to remove the cyclic reference and try again.

VAL00021  Cannot evaluate properties: [0].

Explanation: The expression could not be evaluated.

User response: Check the expression for syntax errors and try again.

VAL00022  Validation failed: Cannot set [0] to [1]. Value must be [2] [3].

Explanation: The property can not have the indicated value. Possible values are listed in the error message.

User response: Edit the property’s expression and try again.

VDS1000  [0]

Explanation: The indicated error occurred while processing the virtual data source.

User response: Resolve the indicated error and try again.

VDS1001  Internal Error.

Explanation: An internal error occurred.

User response: Collect support data by selecting Help > Collect Support Data and forward it to customer support.

VDS1002  No data source was specified for the query.

Explanation: No data source was specified for the query.

User response: Specify a data source for the query.

VDS1003  Configuration directory is not defined.

Explanation: The configuration directory is not defined.

User response: Define the configuration directory.

VDS1004  Source data source not found for virtual table "[0]."

Explanation: The source data source for the indicated virtual table was not found. It is possible that the source data source has been moved or deleted.

User response: Check the location of the source data.
source and copy the source table to the virtual data source again if necessary.

VDS1005 Table is not defined in the query.
Explanation: The table is not defined in the query.
User response: Define the table in the query.

VDS1006 Source table column not found for virtual column "(0)" in the table "(1)."
Explanation: The indicated virtual column does not match any of the source table columns.
User response: Ensure that every column in the virtual table has a corresponding column in the source table.

VDS1007 Only tables and queries can be added to the virtual data source.
Explanation: You attempted to move an object that was not a table or a query to the virtual data source. Only tables and queries may be added to the virtual data source.
User response: Specify a table or query to be moved to the virtual data source.

VDS1008 Cannot find specified tables in data source.
Explanation: The tables that are referenced in the virtual data source can not be found in the source data source. This may be because the source tables were moved or deleted.
User response: Check the location of the source tables in the data source and copy them to the virtual data source again if necessary.

VDS1009 Cannot remove the last column from virtual table.
Explanation: You attempted to remove the last column from the virtual table. A virtual table must have at least one column.
User response: Ensure that the virtual table has at least one column.

VDS1010 Only queries for relational data sources can be added.
Explanation: You attempted to add a query from a multidimensional data source. Only queries for relational data sources can be added to virtual data sources.
User response: Specify queries for relational data sources to be added to the virtual data source.

VDS1011 Source table not found "(0)".
Explanation: The indicated source table was not found. This may be because the source table was moved or deleted.
User response: Check the location of the source table and copy it to the virtual data source again if necessary.

VDS1012 Source query not found "(0)".
Explanation: The indicated source query was not found. This may be because the source query was moved or deleted.
User response: Check the location of the source query and copy it to the virtual data source again if necessary.

VDS1013 Only SQL queries with SELECT statement can be used to create virtual tables.
Explanation: You attempted to create a virtual table using a source query that does not contain a SELECT statement. Only queries with SELECT statements can be used as source queries.
User response: Specify a query that contains a SELECT statement as a source query.

VDS1014 Virtual table cannot be created because source query returns columns with ambiguous names.
Explanation: The virtual table sources a query that returns a result set that has columns with identical names.
User response: Change the result set column names in the source query and try again.

VDS1015 Virtual table cannot be created because source query returns one or more unnamed columns.
Explanation: The source table of a virtual table must not return unnamed columns.
User response: Specify a source query that does not return unnamed columns.

VDS1016 Virtual table cannot be created because source query returns multiple result sets.
Explanation: Virtual tables can only be created using source tables that return single result sets.
User response: Specify a source table that returns a single result set.
Notices

This information was developed for products and services offered in the US. This material may be available from IBM in other languages. However, you may be required to own a copy of the product or product version in that language in order to access it.

IBM may not offer the products, services, or features discussed in this document in other countries. Consult your local IBM representative for information on the products and services currently available in your area. Any reference to an IBM product, program, or service is not intended to state or imply that only that IBM product, program, or service may be used. Any functionally equivalent product, program, or service that does not infringe any IBM intellectual property right may be used instead. However, it is the user's responsibility to evaluate and verify the operation of any non-IBM product, program, or service.

IBM may have patents or pending patent applications covering subject matter described in this document. The furnishing of this document does not give you any license to these patents. You can send license inquiries, in writing, to:

IBM Director of Licensing
IBM Corporation
North Castle Drive, MD-NC119
Armonk, NY 10504-1785
US

For license inquiries regarding double-byte (DBCS) information, contact the IBM Intellectual Property Department in your country or send inquiries, in writing, to:

Intellectual Property Licensing
Legal and Intellectual Property Law
IBM Japan, Ltd.
19-21, Nihonbashi-Hakozakicho, Chuo-ku
Tokyo 103-8510, Japan

INTERNATIONAL BUSINESS MACHINES CORPORATION PROVIDES THIS PUBLICATION "AS IS" WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF NON-INFRINGEMENT, MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. Some jurisdictions do not allow disclaimer of express or implied warranties in certain transactions, therefore, this statement may not apply to you.

This information could include technical inaccuracies or typographical errors. Changes are periodically made to the information herein; these changes will be incorporated in new editions of the publication. IBM may make improvements and/or changes in the product(s) and/or the program(s) described in this publication at any time without notice.

Any references in this information to non-IBM websites are provided for convenience only and do not in any manner serve as an endorsement of those websites. The materials at those websites are not part of the materials for this IBM product and use of those websites is at your own risk.

© Copyright IBM Corp. 1995, 2015
IBM may use or distribute any of the information you supply in any way it believes appropriate without incurring any obligation to you.

Licensees of this program who wish to have information about it for the purpose of enabling: (i) the exchange of information between independently created programs and other programs (including this one) and (ii) the mutual use of the information which has been exchanged, should contact:

IBM Director of Licensing
IBM Corporation
North Castle Drive, MD-NC119
Armonk, NY 10504-1785
US

Such information may be available, subject to appropriate terms and conditions, including in some cases, payment of a fee.

The licensed program described in this document and all licensed material available for it are provided by IBM under terms of the IBM Customer Agreement, IBM International Program License Agreement, or any equivalent agreement between us.

Information concerning non-IBM products was obtained from the suppliers of those products, their published announcements or other publicly available sources. IBM has not tested those products and cannot confirm the accuracy of performance, compatibility or any other claims related to non-IBM products. Questions on the capabilities of non-IBM products should be addressed to the suppliers of those products.

This information contains examples of data and reports used in daily business operations. To illustrate them as completely as possible, the examples include the names of individuals, companies, brands, and products. All of these names are fictitious and any similarity to the names and addresses used by an actual business enterprise is entirely coincidental.

COPYRIGHT LICENSE:

This information contains sample application programs in source language, which illustrate programming techniques on various operating platforms. You may copy, modify, and distribute these sample programs in any form without payment to IBM, for the purposes of developing, using, marketing or distributing application programs conforming to the application programming interface for the operating platform for which the sample programs are written. These examples have not been thoroughly tested under all conditions. IBM, therefore, cannot guarantee or imply reliability, serviceability, or function of these programs. The sample programs are provided "AS IS", without warranty of any kind. IBM shall not be liable for any damages arising out of your use of the sample programs.

Each copy or any portion of these sample programs or any derivative work must include a copyright notice as shown below.

© (your company name) (year).
Portions of this code are derived from IBM Corp. Sample Programs.
© Copyright IBM Corp. _enter the year or years_.
Trademarks

IBM, the IBM logo, and ibm.com® are trademarks or registered trademarks of International Business Machines Corp., registered in many jurisdictions worldwide. Other product and service names might be trademarks of IBM or other companies. A current list of IBM trademarks is available on the web at [http://www.ibm.com/legal/copytrade.shtml](http://www.ibm.com/legal/copytrade.shtml)

Java and all Java-based trademarks and logos are trademarks or registered trademarks of Oracle and/or its affiliates.

Linux is a registered trademark of Linus Torvalds in the United States, other countries, or both.

Microsoft, Windows, Windows NT, and the Windows logo are trademarks of Microsoft Corporation in the United States, other countries, or both.

UNIX is a registered trademark of The Open Group in the United States and other countries.

Other company, product, and service names may be trademarks or service marks of others.

Terms and conditions for product documentation

Permissions for the use of these publications are granted subject to the following terms and conditions:

**Applicability:** These terms and conditions are in addition to any terms of use for the IBM website.

**Personal use:** You may reproduce these publications for your personal, noncommercial use provided that all proprietary notices are preserved. You may not distribute, display or make derivative work of these publications, or any portion thereof, without the express consent of IBM.

**Commercial use:** You may reproduce, distribute and display these publications solely within your enterprise provided that all proprietary notices are preserved. You may not make derivative works of these publications, or reproduce, distribute or display these publications or any portion thereof outside your enterprise, without the express consent of IBM.

**Rights:** Except as expressly granted in this permission, no other permissions, licenses or rights are granted, either express or implied, to the publications or any information, data, software or other intellectual property contained therein.

IBM reserves the right to withdraw the permissions granted herein whenever, in its discretion, the use of the publications is detrimental to its interest or, as determined by IBM, the above instructions are not being properly followed.

You may not download, export or re-export this information except in full compliance with all applicable laws and regulations, including all United States export laws and regulations.

IBM MAKES NO GUARANTEE ABOUT THE CONTENT OF THESE PUBLICATIONS. THE PUBLICATIONS ARE PROVIDED "AS-IS" AND WITHOUT
WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING BUT NOT LIMITED TO IMPLIED WARRANTIES OF MERCHANTABILITY, NON-INFRINGEMENT, AND FITNESS FOR A PARTICULAR PURPOSE.

Privacy policy considerations

IBM Software products, including software as a service solutions, (“Software Offerings”) may use cookies or other technologies to collect product usage information, to help improve the end user experience, to tailor interactions with the end user, or for other purposes. In many cases no personally identifiable information is collected by the Software Offerings. Some of our Software Offerings can help enable you to collect personally identifiable information. If this Software Offering uses cookies to collect personally identifiable information, specific information about this offering’s use of cookies is set forth below.

This Software Offering does not use cookies or other technologies to collect personally identifiable information.

If the configurations deployed for this Software Offering provide you as customer the ability to collect personally identifiable information from end users via cookies and other technologies, you should seek your own legal advice about any laws applicable to such data collection, including any requirements for notice and consent.

Glossary

The glossary provides brief descriptions of product terms.

**accessibility.** Features that help those with physical disabilities, such as restricted mobility or limited vision, use their computer.

**breakpoints.** Color maps that specify the value at which the color changes.

**calculated columns.** Columns of data that you add to the query results.

**classic reports.** Text-based, tabular reports that are generated using query results as the data source and a form template.

**color map.** A color map can be used to vary the color of a visual project object depending on a value associated with the object.

**color sequence.** A color sequence can be used to modify the color properties of a visual project object based on an index value.

**command bar.** An interface that allows you to run procedure commands directly from the DataQuant for Workstation interface.

**connectivity tool.** An interface that allows you to define dependencies between non-query driven objects such as text box objects and label objects. The Connectivity tool is only available when creating visual dashboards.

**cube structure tree.** In the layout format, the Cube Structure tree lists the dimensions and measures that are contained in the cube.

**data sources.** Data sources allow you to create virtual tables that can be edited and manipulated by users without altering the original tables' content. Virtual data sources work by introducing a metadata layer which mediates between an administrator-defined, virtual data source and the underlying data sources that contain the physical tables and views.

**data templates.** Data templates are used to create and control the visual representation of query result set data within layout objects.

**Diagram Query Editor.** When building queries using the Diagram Query editor, you supply table, join, column, sort, and row information and the Diagram Query editor constructs the Structured Query Language (SQL) statements. The Diagram Query editor is not available in DataQuant for WebSphere.

**Draw Query Editor.** Use the Draw Query editor to create a query object automatically based on the query type and tables that you specify.

**drilldown.** A navigation feature that enables your users to click an object and jump to another scene that contains more information that relates specifically to the original object's value. The Drilldown wizard is available for all layout objects that can display query result data.

**edit codes.** Edit codes specify how to format character, date, graphic, numeric, time and timestamp data that will appear in a specific column of a report. You can also create user-defined edit codes.

**embedded scene.** An embedded scene object is a container through which you can view all or a portion of another scene and then navigate to that scene. The scene where you place the embedded scene object is considered the source scene. The destination scene is the embedded scene.

**event actions.** You define event actions to objects to provide scene interactivity and navigation. As a user navigates your dashboard, mouse actions trigger predefined events, such as jumps between scenes, executing of SQL statements, or the launching of other applications.

**Expression Designer.** The Expression Designer is used to specify any property value that is associated with a selected object. The Expression Designer is an alternative way of specifying an object's properties. For each property that can be specified for an object, you are given templates that can help you format the value of that property. In addition, you have more space available to work and view your property value (which can grow quite large if expressions are used).

**fast save.** When saving your query results data using a "fast save" method, all the processing occurs at the database. There is no further passing of data between the DataQuant interface and the database. The DataQuant interface adds SQL to the original query, the query is rerun and the data is saved directly into the specified table(s). Saving query results data using the "fast save" process significantly improves performance when a large amount of data is to be saved.

**file-based repositories.** If you are using DataQuant for Workstation, you can create a file repository connection to set up the information that enables you to connect to a file-based repository that is stored on a
local or network drive. File connections can only be created for legacy, file-based repositories.

**formatting options.** You can customize how the query results will appear in the editor window. You can specify different fonts, colors and text alignments for each column heading and each cell of the column. You can also specify that column and cell formatting be applied based on the results of a conditional expression. You can apply formatting options to entire columns, individual cells, column headings, and summary cells.

**forms.** Forms are considered objects and they can be saved in your repository, in the QMF catalog, or in a file. When you open a form object that has been saved, you are actually running the form object to generate the report. When opened, forms automatically use the currently active query results as the data source.

**form variables.** Form variables are codes that you can insert into text fields to produce information on the report. For example, you can insert a date variable to produce the current date whenever the report is printed.

**global parameter.** A global parameter is available to your entire visual project and to the user of your visual dashboard. You can pass global parameters when you specify object properties or when you create event actions. You also have the option of making global parameters public at runtime so they are accessible to the user.

**global resources.** Global resources can be used by all the pages of a visual report or all the scenes of a visual dashboard. You can use global resources for computing object properties.

**global variables.** Global variables are variables that stay active while the current session of DataQuant is active. This is in contrast to substitution variables that are active only during the execution of an object (query, procedure, form). For objects that use global variables, the value currently defined for the global variable is used.

**grouping and aggregation.** Grouping and aggregation options can be applied to query result columns to organize the result data into logical or summarized groupings. By adding grouping and aggregation you can automatically obtain summary information about your data and display the data more logically. For example you can roll up data in a report by department or average departmental commissions.

**information zooming.** Information zooming is a navigation feature that presents a more detailed view of data as your user zooms in on a scene or a data point in a layout. Every layout object and scene has a default zoom level (100% zoom). You can set up information zooming navigation for whole scenes and for individual layout objects. Information zooming on the layout object allows you to provide more query information on each data point as the user zooms in on a layout object. You use the same query results data for each zoom level. Information zooming on the scene level allows you to display an entirely different layout and query results data for each zoom level.

**JDBC drivers.** DataQuant uses JDBC to connect to all database repositories and data sources. DataQuant does not include JDBC drivers. The location of the JDBC drivers that DataQuant will use to connect to repositories and data sources must be defined. DataQuant administrators are responsible for identifying where each instance of the application interface will find the JDBC drivers.

**job definition files.** Job definition files (jdfx) are legacy DataQuant files that can be imported to set up scheduled jobs.

**job scheduler.** DataQuant has the ability to schedule a procedure object to be executed on a periodic, recurring basis or at a specific date and time. A DataQuant procedure can perform one or more sequential steps, such as run queries, apply report templates, export reports to PDF files, and send reports using email. You can schedule your procedure to run at any given time using your local scheduler or the DataQuant for WebSphere scheduler.

**layout objects.** In a visual dashboard scene, layout objects are used to display query results in visual formats, such as charts, graphs, maps, tables and grids. For most layout objects, you display the results of multiple queries in a single layout. For example, you could create a single XY chart that displays sales figures derived from one query and spending figures derived from another query. You can nest multiple layout objects. When you nest layout objects, you can pass query result information from a higher level layout object to a lower level layout object. Because you have the ability to pass this information, you can use nested layout objects to display more detailed information that relates to a specific data value.

**Layout Properties dialog.** An interface that allows you to set all the options that are available for formatting query results. The Layout Properties dialog uses a tree structure to represent the columns in the query results. Depending on what you have selected in the query results, the Layout Properties tree displays branches for all columns in the query results or for a single column in the query results. You can apply formatting options to entire columns, individual cells, column headings, and summary cells.

**Legacy Visionary world.** You can create a new dashboard from an existing Visionary world that was created using a previous version of QMF Visionary. The Visionary world that you import must have been exported as an XML file using QMF for Visionary Version 8.1, Fix Pack 13 or later.
LOB data. A large object (LOB) is a DB2 for z/OS and DB2 for Linux, UNIX®, and Windows data type that houses nontraditional data such as text, multimedia, image, video, photograph, sound, or any very large data file inside a database table. Retrieving or saving LOB data can consume a substantial amount of resources.

network repositories. A connection to a shared repository using direct network access. Network connectivity permits users to access a shared repository without the need for direct database connectivity from the workstation. Connecting to repositories using a network connection allows for consolidation of data access at a central server and removes the need to distribute JDBC drivers across each desktop running DataQuant for Workstation. This type of connection is not applicable to DataQuant for WebSphere users.

OLAP queries. OLAP queries enable users to dynamically access different dimensions of multidimensional data. Using DataQuant for Workstation and DataQuant for WebSphere, you can create an OLAP query for multidimensional data sources that are queried using MDX and accessed using XMLA.

palette objects. From the Palette view, you can insert any graphical object that is listed into your visual project. All objects from the Palette view can be inserted in a visual dashboard. Not all objects available from the Palette view can be inserted in a visual report. The objects that cannot be inserted in a visual report have interactive properties, and visual reports are not interactive.

personal repositories. Personal repositories serve a single user and are stored in the personnel settings directory of the workstation (server) that is running the DataQuant application interface.

perspectives. An interface used for querying data and formatting results.

procedures. A set of commands that enable you to run queries, print reports, import and export data, as well as perform other functions.

Project Explorer view. The Project Explorer view details in a tree structural all the elements and objects of each visual report and each dashboard.

Prompted Query editor. An interface that allows you to build simple to complex queries. The Prompted Query editor prompts you for the information it needs to build the SQL statement. You can also set sort and row conditions using the Prompted Query editor. (DataQuant for Workstation only)

QMF catalogs. A set of database tables that contain saved objects (queries, procedures and forms); user resource limits and profiles; reports; and other miscellaneous settings and information. QMF catalogs reside on database servers that host a DB2 database.

Query Editor. An interface that allows you to open any database table that is accessible to you in your workspace.

query parameters. Query parameters contain the value that will be sent to the query and used at run time.

regular save. When saving your query results data using a “regular save” method, the interface that you are using (DataQuant for Workstation or DataQuant for WebSphere) saves the retrieved query results to the database using an individual SQL INSERT statement for each row of data. This type of save can be expensive in terms of performance due to the overhead of passing large amounts of data back to the database.

relational query. A query is a request for information from a data source. To request information from a relational data source your query is constructed using SQL statements.

resource limits. DataQuant resource limits control your data source access and resource consumption. You must be connected to the data source in order to view the resource limits that are in effect for your user ID when you access that data source.

scene parameters. A parameter that is available only to a specific scene in a visual dashboard. You can pass scene parameters when you create embedded scenes and event actions.

scheduled jobs. You can schedule your procedure to run at any given time using your local scheduler or the DataQuant for WebSphere scheduler. Procedures that have been scheduled to run at another time are referred to as scheduled jobs.

shared repositories. Shared repositories are located on a database and can be shared by many users. Only DataQuant administrators can create shared repositories.

SQL Query editor. For those with SQL experience, one way of creating a query is to type their own SQL statements in the SQL Query editor. You can write a single SQL statement that will return a single result set or multiple SQL statements that will return multiple result sets. The editor provides coloring support for your SQL statements.

static SQL packages. Several tables in repository storage are used by DataQuant to store processing information. Some of these tables store sensitive information, for example the permissions table. By default all users can access these repository storage tables and make changes to the tables. You can choose to protect the repository storage tables. In protection mode, the repository storage tables are accessed using a
collection of stored procedures or static SQL packages depending on what the database that is hosting the repository storage supports. Users of the repository storage must then be granted permission to run the stored procedures or static SQL packages.

**substitution variables.** Substitution variables are used to enter changing values to a SQL query at run time. This feature enables you to substitute a part of an SQL statement and make it more generic. Substitution variables are active only while the object (query, procedure or form) is running. As a result, only one object can access the substitution variable. The variable will not exist after the object is executed.

**Table Editor.** An interface that allows you to open any database table that is accessible to you in your workspace.

**Table Viewer.** An interface that allows you to open any database table that is accessible to you in your workspace.

**translation tables.** Translation tables enable dashboard designers to publish a single visual dashboard that may be viewed in multiple languages. When content developers create a dashboard, button captions, labels, tool tips and other embedded text is entered directly into the property values of the dashboard objects. Using translation tables, content developers can supply alternate versions of this entered text, to be displayed when the user is in a different locale. For example, a Japanese developer may elect to author a new dashboard in Japanese but then also include translation tables for German and French, each of which will be used when the dashboard is viewed under German and French locales.

**usage codes.** Usage codes provide summary information about the data in a column. For example, usage codes can provide total summary information at the end of a column, or partial summaries at control breaks in a table. The usage codes available depend on the data in the column and the type of summary.

**user preferences.** The Preferences dialog is used to set user preferences that will apply to specific functions in the current DataQuant application interface session. The Preferences dialog consists of two panes. The left pane displays a hierarchical tree to organize the preferences and the right pane displays the pages of actual preference values.

**variables bar.** You can use the Variables bar to specify a property value. The Variables bar is located in the menu bar. From the Variables bar you can choose a global or local parameter that has been defined for the visual project and a column name from a set of query results.

**views.** DataQuant views support editors and provide alternative presentations as well as ways to navigate the information in your window. Each view displays as a pane in the main workstation window. Each perspective of DataQuant initially displays the views that are most commonly used to perform specific functions.

**visual dashboards.** Visual dashboards present interactive or persistent data obtained from querying multiple heterogeneous data sources across the enterprise. Visual dashboards present data in a scene format using a wide variety of graphics including charts, graphs, maps and user interface widgets. Data driven graphical objects can be easily linked so that user selections will trigger unique data displays. Content developers can create a visual dashboard that can be viewed by multiple users with either DataQuant for Workstation or DataQuant for WebSphere.

**visual designer.** You can quickly design visual reports and dashboards using the intuitive Visual Designer perspective that includes an editor that presents both design and runtime views of the project; the Project Explorer view which details the structural content of each visual report and each dashboard; and the Palette, Properties, Events and Output views which support the Visual Designer editor and assist in creating visual reports and dashboards.

**visual reports.** Visual reports are page-based, printable reports that include both formatted text and graphics to display persistent data to a wide variety of users. Visual reports can also contain data driven graphics (such as maps and charts) inserted in different sections (such as the headers or footers) of the report. Each of the data driven graphics can present data from multiple queries that are run across the enterprise.

**web service repositories.** You create a Web service repository connection to set up the information that enables you to connect to a shared repository through DataQuant for WebSphere web services API using a HTTP or HTTPS connection. Web-based connectivity permits users to access a shared repository without the need for direct database connectivity from the workstation. Connecting to repositories using a web-based connection allows for consolidation of data access at a central server and removes the need to distribute JDBC drivers across each desktop running DataQuant for Workstation. This type of connection does not apply to DataQuant for WebSphere.

**workspaces.** All of the data sources and objects that you can access are contained in one or more workspaces that have been pre-populated for you by the DataQuant administrator. Each workspace to which you have access is listed in the Workspaces view. From the Workspaces view, you can perform the majority of query and reporting functions.

**zoom.** If you are using DataQuant for Workstation, you can use the Zoom option to view in hexadecimal or binary the contents of a cell in the query results.
Index

Special characters
/Batch 452
/IDisplay 452
/IObject:“ObjectName” 452
/IPassword:password 452
/IServer:servername 452
/IUserID:userID 452
/RConnection:connectionname 452
/RDBUser:dbuserID 452
/RObject:pathtoobject 452
/RPassword:password 452
/RUserID: userID 452
/RRun 452
/RUser:userID 452
&an 175
&CALCid 175
&COUNT 175
&DATE 175
&Global Variables 175
&HTML Variables 175
&n 175
&PAGE 175
&RROW 175
&TIME 175
“&variablename=variablevalue” 452

A
about 415, 416
about extended layout objects 375
about resource limits 455
accessibility 467
assistive technology 468
keyboard equivalents 467
keyboard navigation 467
navigation in WebSphere 470
navigation in Workstation 468
operating system 467
ACROSS 176
actions
action groups 151, 152
execute object 139
export query 140
export query to mail 141
export quick report 145
export quick report to mail 144
export visual report 147
export visual report to mail 146
print query 143
resource manipulation 148
copy 148
send mail 149
Ad hoc reports
creating 247
Add Calculated Column 132
Add to Favorites 22
Add to Startup 22
adding an Execute JavaScript event
how to 277
adding dictionaries 419
adding objects 345

B
best fit models 201
binary 130
binding resource limits 460
Boolean 257
Border 266
Break Footing 175
Break Heading 175
Breakn 176
BREAKnX 176
breakpoints 390
Breaks formatting options
Blank lines after footing 165
Blank lines after heading 165
Blank lines before footing 165
Blank lines before heading 165
Break footing text 165
Break heading text 165
New page for break 165
New page for footing 165
Put break summary at line 165
Repeat detail heading 165

C
cache resource limits 465
CALCid 176
calculated columns 109, 110, 111, 112, 301
calculated expressions 316
Calculation Expression 175
Calculations formatting options
Edit 168
Expression 168
ID 168
Pass nulls? 168

© Copyright IBM Corp. 1995, 2015
Calculations formatting options (continued)
  Width 168
  canvas
    editing linked objects 352
    linked objects 352
  Canvas 343
    types 342
  Canvas mode 309
  Category Axis 229
  category chart format options 220
  CCSID 130
  Change favorite name 28
  Change Startup Name 28
  character data edit codes 439
  chart area chart format options 221
  charts 217, 227
    creating 217
    creating from query results grid 226
    creating in Display editor 227
    editing 230
    specifying chart format 219
    specifying chart types 218
    specifying data series 218
  Cheat Sheets view 3
  classic reports 159
  client area chart format options 226
  Close dialog 291
  Cloudant databases
    adding Cloudant documents 84
    editing Cloudant documents 84
    managing data 83
    removing Cloudant documents 84
    viewing Cloudant documents 83
  Cloudant indexes 75
    creating search indexes 77
    creating secondary indexes 76
    indexes for Cloudant native queries 74
    working with secondary and search indexes 75
  Cloudant queries 71
    creating 72
    creating in Cloudant Native Query editor 74
    creating in Cloudant Query editor 73
  Color 257
  color maps 388, 390
    creating 390
    using 391
  color sequences 388
    creating 389
    using 389
  Column Expression 175
  Columns formatting options
    Edit 168
    Heading 168
    Indent 168
    Num 168
    Seq 168
    Usage 168
    Width 168
  Columns with aggregates 228
  command bar 453
  command line 451
    parameters 452
  comments, sending to IBM xv
  compile
    visual dashboard 302
    visual report 302
  compound condition filter 107
  Condition 175
  conditional grouping 68
  Conditions formatting options
    Expression 170
    ID 170
    Pass nulls 170
    configuring 416
  Connectivity 317
  convert
  convert CSV 127
  convert DBF 127
  convert IXF 127
  convert TXT 127
  convert XML 127
  Convert to HTML 200
  Convert to Visual Report 200
  Copy 22
  Count 132
  COUNT 176
  CPCT 176
  Create
    JavaScript Tables 407
    Create Cyclicality 201
    Create File 287
    Create New Visual Dashboard wizard
      tabs
        Local 252
        Predefined 252
        Repository 252
    Create New Visual Report wizard
      Local tab 237
      Predefined tab 237
      Repository tab 237
    Create Performance 201
    Create Seasonality 201
    Create Trend 201
    creating 418, 420
    crosstab queries 64
    CSUM 176

548 Getting Started with DataQuant
cube data 57
Cube Structure tree 57, 58
cumulative forecast error 210
Cumulative Percentage of Group 132
Cumulative Percentage of Total 132
Cumulative Sum 132
curve fitting 209
Curve fitting methods 201
Custom 317
custom functions 112
customizing perspectives 8
data source connections 255
Data Source Connections view 3
Data Source Object 200
data source options
  embedded 203
  linked 203
data sources 178
  listing objects 161
  selecting new 87
data symbol objects
  Candlestick 335
  EventBand 335
  HorizontalLabelBar 335
  HorizontalValueBar 335
  Marker 335
  PriceIntervalBar 335
  ThresholdBand 335
  VerticalLabelBar 335
  VerticalValueBar 335
Data Symbols 317
data templates 380
  visibility 381
data types 315
Database folder 21
DataQuant Calculator view 3
date data edit codes 440
date parameters
  date column 203
  from 203
  number of future periods 203
  period 203
DateTime 257
DateTimeSet 257
DB2 for z/OS 1
Debugging visual dashboards 405
Default editor 78
Delete 22
Design mode 309
detail 99
Detail Block 175
Detail Heading 175
Details formatting options
  Blank lines after block 170
  Detail block text 170
  Detail heading text 170
  Enable 170
  Include column headings with detail headings 170
  Keep block on page 170
  New page for detail block 170
  Put tabular data at line 170
  Repeat detail heading 170
dialog templates (continued)
  border 293
  content paddings 294
  Nine Patch 294
  scene background 295
  shadow 295
  title background 295
  title font 296
  title frame 296
directly importing an IXF file into a database table 126
Display editor 227
Display editor panes
  Display modes 227
  Layout structure 227
  Result set 227
Display Excel Sheet 132
display modes 227, 228
editing chart format
  axes 221
  category 220
  chart area 221
  client area 226
  gridlines 223
  labels 223
  legend 225
  plot area 225
  range 224
  series 220
  title 225
  value(Y)series 221
  X-Axis 222
  Y-Axis 222
Display modes pane 227, 228
Display Report 132
Display Report wizard 159
display settings
  Columns 320
  Details 320
  Icons Only 320
  List 320
double exponential smoothing 207
Double Exponential Smoothing 201
Draw Query editor 48
  listing data source tables 49
Draw Query wizard 30
Drill Down 132
Drill Up 132
drilldown 260
Drilldown wizard 260
driver class name 14
DSQAO 433
DSQCP 438
DSQDC 437
DSQEC 435
DSQQW 428
Dynamart Filter Wizard 107
dynamic SQL 1
Eclipse 1
Edit
  JavaScript Tables 414
edit codes 439
  character data 439
  custom codes for classic reports 445
date data 440
edit codes (continued)
  graphic data 441
  metadata 450
  numeric data 441
  samples 446
  time data 443
  timestamp data 444
  user defined 445
  editing 421, 422
  editing database tables 83
  editor window 309
editors
  Cloudant Query 30
  Prompted Query 30
  Query Builder 30
  SQL 30
email
  query results 116
  embedded scenes 266, 267
  parameterized 267
End Session 285
event actions 269
  assigning 269
  Close dialog 298
  Create File 287
End Session 285
  Execute JavaScript 277
  Execute procedure 269, 276
  Execute shell command 269, 280
  Execute SQL statement 269, 274
  Export to excel 282
  Export to Excel 269
  Export to PDF 285
  Invalidate Query Cache 289
  Jump to new location 269, 271
  Launch LOB 290
  Load Cookie 300
  Navigate back 269, 274
  Navigate to an embedded scene 273
  Navigate to embedded scene 269
Open URL 299
  Print scene 269, 277
  Refresh Component 289
  Refresh geolocation 301
  Reload scene 284
  Save Cookie 300
  Send Email 278
  Send Event to Embedded Scene 279
  Set focus 284
  Set values 269, 272
  Show dialog 297
  Show message 269, 281
EventInterval 322
events
  about 273
  ExternalEvent events 273
Events view 3, 317
  Execute JavaScript 269, 277
  Execute procedure 269, 276
  Execute shell command 269, 322
  Execute SQL statement 269, 322
expiration schedules 388, 397
daily 399
  hourly 398
  monthly 399
  specific dates 400
  weekly 399
Explore 22
export
  chart 230
  forecast 214
  query results 116
  database 118
  file 119
  file export options 121
  reports 162
  Word Map 424
Export 132
Export images 308
export query operation 140
export query to mail operation 141
export quick report operation 145
export quick report to email operation 144
Export to excel 282
Export to Excel 269
Export to PDF 285
export visual report operation 147
export visual report to mail operation 146
expression designer 107, 313
expressions 314
  column names 314
  constants 314
  elements 314
  examples 316
  formatting symbols 314
  functions 314
  operators 314
  syntax rules 315
external code 410
ExternalEvent events 273
ExternalEventID 279

F
fast save 136
Favorites folder 28
feedback, sending to IBM xv
file formats
  HTML 162
  PDF 162
text 162
FilePath 257
filter page 57
Filter tab 60
filtering query results 107
Final formatting options
  Blank lines before text 173
  New page for final text 173
  Put final summary at line 173
  Text 173
Final Text 175
Find 132
First 132
FIRST 176
First Sort Rule 134
Flash 309
Font 132
FontName 257
forecast chart properties 213
forecast cyclicality 213
forecast grid properties 213
forecast performance
  cumulative forecast error 210
  mean absolute deviation 210
forecast performance (continued)
  mean absolute percent error  210
  mean square error  210
  tracking signal  210
forecast seasonality
  global average  212
  moving average  212
forecast trends
  global trend  211
  linear trend  211
  local trend  211
forecasting models options
  Curve fitting  206, 209
  Double exponential smoothing  206, 207
  Holt-Winters method  206, 208
  Multiplicative decomposition  206, 208
  Neural network  206, 210
  Polynomial regression  206, 209
  Simple moving average  206
  Single exponential smoothing  206, 207
  Weighted moving average  206, 207
forecasts  203
  chart properties  213
  cyclicity  213
  data source options  203
  date parameters  203
  forecasting models options  206, 207, 208, 209, 210
  grid properties  213
  grouping hierarchy options  204
  overview  202
  performance  210
  seasonality  212
  strategy options  205
  trends  211
Form menu commands
  Convert to HTML  200
  Convert to Visual Report  200
  Data Source Object  200
  Set Data Source  200
  Set Font  200
  Set User Information  200
Form structure tree
  Breaks  163
  Calculations  163
  Columns  163
  Conditions  163
  Details  163
  Final  163
  Main  163
  Options  163
  Page  163
form variables
  &an  175
  &CALCId  175
  &COUNT  175
  &DATE  175
  &Global Variables  175
  &HTML Variables  175
  &n  175
  &PAGE  175
  &ROW  175
  &TIME  175
Format  132
formatting options (continued)
  breaks (continued)
    Blank lines before footing  165
    Blank lines before heading  165
    Break footing text  165
    Break heading text  165
    New page for break  165
    New page for footing  165
    Put break summary at line  165
    Repeat detail heading  165
calculations
  Edit  168
  Expression  168
  ID  168
  Pass nulls?  168
  Width  168
columns
  Edit  168
  Heading  168
  Indent  168
  Num  168
  Seq  168
  Usage  168
  Width  168
conditions
  Expression  170
  ID  170
  Pass nulls  170
details
  Blank lines after block  170
  Blank lines after footing  170
  Blank lines after heading  170
  Break footing text  170
  Break heading text  170
  New page for block  170
  New page for detail block  170
  New page for footing  170
  New page for heading  170
  Repeat detail heading  170
final
  Blank lines before text  173
  Blank lines before footing  173
  Blank lines before header  173
  New page for final text  173
  New page for final footing  173
  New page for final header  173
  Put final summary at line  173
  Repeat detail heading  173
main
  Edit  163
  Heading  163
  Indent  163
  Seq  163
  Total width of report columns  163
  Usage  163
  Width  163
options
  Across heading separator  172
  Across summary column  172
  Automatic reordering of report columns  172
  Automatic reordering of report headings  172
  Break summary separator  172
  Column heading separator  172
  Column wrapped lines kept on a page  172
  Default break text (*)  172
  Detail spacing  172
  Final summary separator  172
  Include column headings with detail headings  170
  Include column headings with summary headings  172
  Include summary column with detail headings  172
  Include summary column with summary headings  172
  Line wrapping width  172
  Number of fixed columns in report  172
  Number of fixed columns in summary  172
  Outlining for break columns  172
  Outlining for column headings  172
  Place LOB contents inline  172
  Put detail text at line  170
  Put final detail at line  173
  Repeat detail heading  173
  Repeat summary at line  173
  Text  173

Index  551
formatting options (continued)
  options (continued)
    Report text line width 172
    page
      Blank lines after footing 174
      Blank lines after heading 174
      Blank lines before footing 174
      Blank lines before heading 174
    Page footing text 174
    Page heading text 174
forms 162
  designing 163
  saving with data source 179
  saving without data source 179
free style condition filter 108
Frequently Used 317
From data source 160
From file 160
From open object 160
full outer join 38, 44
functions.js 111

G
General preferences 9
getting started 1
global average 212
global parameters 394
  adding 396
  ExternalEventID 395
  SendExternalEvent 395
  Translation Table 395
  using 397
  ViewerHeight 395
  ViewerWidth 395
  ViewerX 395
  ViewerY 395
  ViewerZoom 395
Global parameters 312
global resources
  color maps 388, 390
    creating 390
    using 391
  color sequences 388
    creating 389
    using 389
  expiration schedules 388, 397
    daily 399
    hourly 398
    monthly 399
    specific dates 400
    weekly 399
  global parameters 394, 395
    adding 396
    using 397
  images 388
  parameters 388
  security lists 388, 392
  stock images 393
    creating 393
    using 394
  translation tables 388, 400
  switching 402
global trend 211
global variables 11, 427, 428
  DSQAO 433
global variables (continued)
  DSQCP 438
  DSQDC 437
  DSQEC 435
  DSQQW 428
  system 12, 428
  user 11, 427
Go to 132
Google Maps 382
  GoogleMap object 382
  GoogleMapMarker object 383, 384
  GoogleMapObject object 386
  GoogleMapPolygon object 385
  GoogleMapPolyline object 385
Grab 317
grand total 99
graphic data edit codes 441
gridlines chart format options 223
GROUP 176
grouping and aggregation 114, 229
  fields 114
Grouping and Aggregation
  Average 132
  Count 132
  Cumulative Percentage of Group 132
  Cumulative Percentage of Total 132
  Cumulative Sum 132
  First 132
  Last 132
  Maximum 132
  Minimum 132
  No Aggregation 132
  Percentage of Group 132
  Percentage of Total 132
  Side Group 132
  Standard Deviation 132
  Sum 132
  Top Group 132
  grouping hierarchy options
    available columns 204
    selected columns 204

H
header 99
help content 13
Help preferences 12
Help view 3
hexadecimal 130
Hierarchical filter 60
highlighting conventions xiii
Holt-Winters method 201, 208
host 13
HP-UX 1
HTML 309
HTML tables 156
HTML variable 167

I
ICatalogPassword:catalogpassword 452
ICatalogUserID:cataloguserID 452
icons
  Create Cyclicility 201
  Create Performance 201
  Create Seasonality 201
icons (continued)
  Create Trend 201
  Run Query 201
  Show Forecast Outline Tree 201
IMAGEn 167
images 388
import
  CSV 123
    result set structure 124
    separator options 123
  legacy Visionary world 254
  legacy visual reports 240
  table data 126
  tables 126
  TXT 123
    result set structure 124
    separator options 123
import CSV 123
    result set structure 124
    separator options 123
import TXT 123
    result set structure 124
    separator options 123
importing data contained in ixf files 125
information zooming 261
layout objects 261
  scenes 263
Informix 1
inner join 38, 44
Integer 257
Invalidate Query Cache 289
iterator 152

J
JARS 14
JavaScript 408, 409, 410
  functions 305
    module 305
  JavaScript functions 110, 111, 112
  JavaScript modules 112
  JavaScript Modules 306
  JavaScript Tables 407, 408, 411
JDBC 1
  JDBC drivers 1
  JDBC libraries 14
  JDBC preferences 14
JDF 94
job definition files 94
job files 94
join queries 63
Jump to new location 269, 271, 322

K
knowledge base 416
knowledge base dictionaries 418

L
labels chart format options 223
Last 132
LAST 176
Launch LOB 290
Layers view 3, 387
layout objects
  Bar chart 353
  BarChart 357
    extended 358
  OLAP 359
  Candlestick chart 353
  CandlestickChart 364
  Cluster graph 353
  ClusterGraph 366
  Column chart 353
  ColumnChart 357
    extended 358
  OLAP 359
data symbols 377
  graphic objects 378
data templates 379
  new 379
default connectors 377
Dial 373
editing a solid chart 374
Event Band chart 353
EventBandChart 365
  extended 375
extending functionality 375
Grid 353, 357
Horizon 353
information zooming 261
Linear map 353
LinearMaps 366
Matrix 353, 368
multiple query results 378
Multivariate chart 353
MultivariateChart 362
nesting 381
OLAP data 376
Organization chart 353
OrganizationChart 369
Pie chart 353
PieChart 360
  extended 361
  OLAP 361
queries 375
query parameters 375
Scatter chart 353
ScatterChart 363
Simple form 353
SimpleForm 367
Solid chart 353, 374
Spiral 353, 368
Stock chart 353
StockChart 363
Table 353, 354
  populated 355
  simple 354
  summary row formatting 356
  summary rows 356
Timeline chart 353
TimelineChart 365
Tree chart 353
TreeChart 369
XY chart 353
XYChart 363
Layout page 57
layout properties
  Conditional 99
  Font 99
  Format 99
General 99

Layout Properties dialog 99

conditional formatting options 106
font fields 103
font formatting options 102
format fields 104
formatting options 103
general column formatting 101

Layout structure pane 227, 229
Layout Structure tree 114
Layout tab 58
Layout view 59
Layouts 317

left outer join 38, 44
legacy Visionary worlds
importing 254

legend chart format options 225
Level of Detail 261, 263
limits resource limits 456
linear trend 211

LINKn 167
links
non-IBM Web sites 541
Linux 1
LITERAL 257
literals 316
Load Cookie 300
LOB data 129
LOB options resource limit
overriding 130
LOB options resource limits 463
LOBs preferences 15
Local parameters 312
local trend 211
Log preferences 16

MAILTON 167

Main formatting options
Edit 163
Heading 163
Indent 163
Seq 163
Total width of report columns 163
Usage 163
Width 163
manage data files in dynamarts 128
manage in dynamart
CSV 128
DBF 128
IXF 128
TXT 128
XML 128
map 301
Mapping table 129
Maximum 132
MAXIMUM 176
MDX page 57
mean absolute deviation 210
mean absolute percent error 210
mean square error 210
measures 58, 59

menu commands
Personal view 28
Workspace 22

messages
methods for accessing xv
metadata edit codes 450
method
appendData() 409
eraseData() 409
getTag() 409
include() 410
log() 410
replaceData() 408
setTag() 409

Microsoft Windows 1
Minimum 132
MINIMUM 176
MinimumZoom 261, 263
Miscellaneous 317
moving average 212
Moving Average 201
multiple result sets 131
multiplicative decomposition 208
Multiplicative Decomposition method 201

name filter 24
Navigate back 269, 274, 322
Navigate to an embedded scene 273
Navigate to embedded scene 269
neural network 210
Neural Networks 201
New 22
No Aggregation 132
normalize queries 65
notices
legal 539
Number 257
NumberSet 257
numeric data edit codes 441

object list resource limits 465
object name 24
object owner 24
object tracking resource limits 461
ObjectName 452
objects
data source 161
palette 319
OLAP folder 21
OLAP queries
building offline 58
building online 59
filtering cube data 60
modifying 58
opening 58
retrieving cube data 57

OMIT 176
Open 22
Open From 22
open perspective 8
Open URL 299
opening 415
Section 125

Options formatting options

- Across heading separator 172
- Across summary column 172
- Automatic reordering of report columns 172
- Break summary separator 172
- Column heading separator 172
- Column wrapped lines kept on a page 172
- Default break text (*) 172
- Detail spacing 172
- Final summary separator 172
- Function name in column heading when grouping 172
- Line wrapping width 172
- Number of fixed columns in report 172
- Outlining for break columns 172
- Page renumbering at the highest break level 172
- Place LOB contents inline 172
- Report text line width 172

Options resource limits 458

Oracle 1

Outline view 3

Output view 3

overview 1

Palette view (continued)

- Timer 322
- parameter hints 35
- parameter types
  - Boolean 257
  - Color 257
  - DateTime 257
  - DateTimeSet 257
  - FilePath 257
  - FontName 257
  - Integer 257
  - Literal 257
  - Number 257
  - NumberSet 257
  - Percentage 257
  - Point 257
  - PointSet 257
  - PolySet 257
  - Text 257
  - TextSet 257
  - parameters 388
  - global 394
  - query 394
  - scene 394
- Paste 22
- Paste Link 22
- path 13
- PCT 176
- PDF 309
- Percentage 257
- Percentage of Group 132
- Percentage of Total 132
- Personal view 3, 28
- Personal view menu commands
  - Change favorite name 28
  - Change Startup Name 28
  - Remove from Favorites 28
  - Remove from Startup 28

Perspectives 6

- Administrator 2, 3
  - closing 7
  - configuring 8
- QMF 3
  - resetting 8
- User 3, 6
- Visual Designer 3, 237, 307

Plot area chart format options 225

- Point 257
- PointSet 257
- polynomial regression 209
- PolySet 257

Port 13

Preferences

- Appearance 9, 10
- Dashboard Runtime 9
- General 9
- Global Variables 9
- Help 9, 12
- JDBC 14
- JDBC Libraries 9
- LOBs 9, 15
- Log 9, 16
- Server-Side File System 18
- SMTP 17
- Third Party Libraries 19
- Third-Party Libraries 9
- user 9

Index 555
preferences (continued)
Visual Designer 9, 20
Preview mode 309
primitive objects
  Alignment panel 324
  Arrow 324
  Double Arrow 324
  Ellipse 324
  Label 324
  Line 324
  MultiPolygon 324
  MultiPolyline 324
  Picture 324
  Polygon 324
  Polyline 324
  Rectangle 324
  Round Rectangle 324
Primitives 317
print options 135, 214, 231, 454
print query operation 143
Print scene 269, 277, 322
procedures 93
  add JavaScript conditions 97
  creating 93
  edit flowcharts 96
  flowcharts 95
  JavaScript 95
Profiling visual dashboards
  Analyzer view 403
  Profiler view 403
Progress view 3
Project Explorer 308, 309
Project Explorer folders
  Connections 235, 249
 Globals 235, 249
  Main Page 235
  Queries 235, 249
  Scenes 249
Project Explorer view 3, 235
prompt
  shared prompts 56
  simple prompts 50
  SQL prompts 50
prompt hierarchies 50
  adding to queries 54
  creating 53
  embedding to queries 55
  working with 53
prompt values
  saving 52
Prompted Query editor
  adding tables 37
    columns 35
    join conditions 35
    joining columns 38
    joining tables 38
    query results 39, 45
    row conditions 35, 41
    sort conditions 35, 40
    tables 35
properties 311
  data types 315
  expression designer 313
  expressions 314
  elements 314
  examples 316
  syntax rules 315
properties (continued)
  groups 312
  literal values 316
Properties 22
Properties view 3, 99, 311
Q
QM Catalog folder 21
QM perspective 3
queries
  analytical 29, 30
  creating 31
  editors 30
  OLAP
    building offline 58
    building online 59
    filtering cube data 60
    modifying 58
    opening 58
    retrieving cube data 57
  profiling 78
  relational 29
  specifying in visual dashboards 304
  specifying in visual reports 304
  standard 29
  visual 29, 30, 33
Query Builder
  add columns 39, 45
  Query Builder editor
    add tables 43
    fields 43, 44
    global row conditions 43, 47
    join conditions 44
    SQL Editor 43, 48
    tables 43
Query Editor 78
query environment 451
query parameters 394
query results 99
  calculated columns 109
  categorization 112
  adding conformance rules 113
  adding expressions 113
  contents of a cell 130
  email 116
  exporting 116
  exporting to a database 118
  exporting to a file 119
  exporting to Excel 119
  file export options 121
  filtering 107
    compound condition 107
    free style condition 108
  formatting 99
  grouping and aggregation 114
  LOB data 129
  sorting 134
Query Structure tree 57, 58
quick reports 187, 196
  changing data source 198
  column headers 183
  create 186
  creating 180
  creating column groups 184
  data format options 195
quick reports (continued)
detail columns 180
details columns formatting 192
displaying details columns 180
displaying grouping columns 180
editing 197
editing data source 198
grouping columns 180
footer 191
header 191
grouping columns formatting 190
individual column formatting 190
modifying 199
report settings formatting 182
working with 179

R
range chart format options 224
Recently Used folder 28
REFn 167
Refresh 22
Refresh Component 289
regular filter 60
regular save 136
relational queries 29
Reload scene 284
remote server 13
Remove from Favorites 28
Remove from Startup 28
Rename 22
Report Center resource limits 464
reports 159
classic
data source 160
designing forms 163
forms 162
saving 178
visual 233
repository connection settings 1
Repository Connections view 3
requirements 418
Reset All 132
Reset Formatting 132
resetting perspectives 8
resource limits 455
binding 460
cache 465
limits 456
LOB 130
LOB options 463
object list 465
object tracking 461
options 458
Report Center 464
save data 460
SQL verbs 457
timeouts 456
viewing 454
resource manipulation
copy 148
send mail 149
Results set pane 227, 228
results
exporting 424
Results menu commands
Add Calculated Column 132

Results menu commands (continued)
AutoFit 132
Display Excel Sheet 132
Display Report 132
Drill Down 132
Drill Up 132
Export 132
Find 132
Font 132
Format 132
Go to 132
Grouping and Aggregation 132
Reset All 132
Reset Formatting 132
Retrieve All 132
Retrieve All LOB Values 132
Show Summaries 132
Show Total Summaries 132
Sort 132
results page 57
Results tab 58, 59, 60
Retrieve All 132
Retrieve All LOB Values 132
REXX Console view 93
right outer join 38, 44
RSBHttPRequest class 410
Run 22
Run Query 201
Runtime mode 309

S
Save Cookie 300
save data objects as dynamarts 127
save data resource limits 460
save methods
fast 136
regular 136
scene parameters 394
scene views 258
SceneCenter 266
scenes 256, 263
schedule types
always expires 244
at specific date 244
daily 244
monthly 244
never expires 244
weekly 244
scheduled tasks 139
action groups 151, 152, 153, 154
actions 139, 148, 151, 152, 153, 154
execute object 139
resource manipulation 148
scheduling 155
Search view 3
Second Sort Rule 134
security lists 388, 392
Select 317
Send Email 278
Send Event to Embedded Scene 279
series chart format options 220
Server-Side File System 9
preferences 18, 19
service information xiii
Set Data Source 200
Set focus 284
**Index**

**viewpoints**
- modifying 265
- navigation 264

**views** 6, 415
- Analyzer 403
- Cheat Sheets 3
- Data Source Connections 3
- DataQuant Calculator 3
- Events 3, 317
- Expressions 405
- Help 3
- Layers 3, 387
- Layout 59
- opening 7
- Outline 3
- Output 3
- Palette 3, 307, 317
- Personal 3, 28
- positioning 8
- Profiler 404
- Progress 3
- Project Explorer 3, 235, 307, 308, 309
- Properties 3, 99, 307, 311
- Repository Connections 3
- REXX Console 93
- Search 3
- Workspaces 3, 21

**views and perspectives** 6

**visual applications** 233

**visual dashboard actions**
- Show dialog 297

**visual dashboard templates** 253

**visual dashboards** (continued)
- parameterized 267
- event actions 269
- information zooming 261
- layout objects 261
- scenes 263
- modal dialog 291, 292, 293, 294, 295, 296
- navigation features 260
- open queries 406
- planning 251
- queries 241, 304
- read-only 302
- renaming 256
- scenes 256
- parameters 257
- specifying data source connections 240, 303
- storyboards 251
- template category 235
- templates 234, 253
- viewpoint navigation 264, 265

**visual designer** 307

**visual designer preferences** 20

**Visual Designer perspective** 3, 237

**visual projects** 308

**visual report**
- read-only 302

**visual report main pages**
- Detail 242
- Detail Footer 242
- Detail Header 242
- Page Footer 242
- Page Header 242
- Report Footer 242
- Report Header 242

**visual reports** 159, 233, 235

**visual reports fixed pages**
- Fixed1 243
- Fixed2 243
- PageFootering 243
- PageHeading 243

**W**

- weighted moving average 207
- Weighted Moving Average 201
workspace properties
  Connection Parameters  25
  LDAP  25
  Login mapping  25
  Plug-ins  25
  Resource Limits  25
  Security  25
workspaces
  adding a new workspace  28
  adding content  26
  creating  27
  discovering contents  21
  filtering data source objects  24
  folders  23
  linking to objects
    repository  25
    workspace  24
  moving objects  26
  properties  25
  refreshing contents  26
Workspaces view  3, 21
Workspaces view menu commands
  Add to Favorites  22
  Add to Startup  22
  Copy  22
  Delete  22
  Explore  22
  New  22
  Open  22
  Open From  22
  Paste  22
  Paste Link  22
  Properties  22
  Refresh  22
  Rename  22
  Run  22
  Set Name Filter  22
  Set User Information  22
  Show in Repository  22

X
  X-Axis chart format options  222
  Xlocation  264
XMLA connectivity  1

Y
  Y-Axis chart format options  222
  Ylocation  264

Z
  zoom  130
  ZoomPct  266